

NEAR EAST UNIVERSITY

Faculty of Engineering



Department of Electrical & Electronic Engineering

GREENHOUSE AUTOMATION

**Graduation Project
EE-400**

Students: VOLKAN KAVAKLIOĞLU(20010566)

ERTUĞ GÜNAY (20010587)

Supervisor: Asst. Professor Dr. Özgür C. Özerdem

Nicosia - 2008

ACKNOWLEDGEMENTS

First I would like to thank to my supervisor Assist.Prof.Dr.Özgür C. Özerdem for his great advice and recommendation for finishing my Project properly also,teaching and guiding me in other lectures.

I am greatly indepted my family for their endless support from my starting day in my educational life until today.

ABSTRACT

A new generation of climate, irrigation and nutrition control will employ crop sensors and models. Feed-forward controllers anticipate the effects of disturbances on the greenhouse climate and take corrective action before they are allowed to occur. Greenhouse models can be used to predict the effects of disturbances. By using a crop model to estimate the benefits to the crop and a greenhouse model to estimate the costs, optimum setpoints can be generated. The reliability of model-based control is significantly enhanced when feedback on the crop's status and growth rate are added. For this purpose, crop sensors need to be developed. Sensor data combined with intelligent algorithms, collectively called 'soft sensors', represent a promising way of obtaining additional information on the growth process. Crop monitoring can also be used as an early warning system (by comparing sensor measurements with reference data) and so help to limit the consequences of human error or technical failure. Optimal controllers use a model-based economic assessment to determine the optimum values for various processes and resource input levels. Optimal control will first be introduced as decision support systems at crop process level.

TABLE OF CONTENTS

| | |
|----------------------------------------------|----------|
| INTRODUCTION | 1 |
| 1. INFORMATION ABOUT PLC | 1 |
| 1.1 PLC History | 1 |
| 1.2 Hardware components of PLC system | 3 |
| 1.3 Central processing unit(CPU) | 3 |
| 1.4 System busses | 3 |
| 1.5 Memory | 4 |
| 1.6 I/Q sections | 4 |
| 1.7 Power supply | 4 |
| 1.2 PLC OPERATIONS | 4 |
| 1.2.1 Input relays | 4 |
| 1.2.2 Internal utility relays | 5 |
| 1.2.3 Counters | 5 |
| 1.2.4 Timers | 5 |
| 1.2.5 Output relays | 5 |
| 1.2.6 Data storage | 5 |
| 1.3 PLC COMMUNICATIONS | 6 |
| 1.3.1 Extension modules | 6 |
| 1.3.2 Remote PLC's | 6 |
| 1.3.3 Cables | 6 |
| 1.3.4 Parallel communications | 6 |
| 1.3.5 Parallel standards | 7 |
| 1.3.6 Serial communication | 7 |
| 1.4 PLC PROGRAMMING | 7 |
| 1.4.1 Programming languages | 7 |
| 1.4.2 Programming devices | 7 |
| 1.4.3 Ladder logic | 8 |
| 1.4.4 Ladder diagram features | 8 |
| 1.5 PLC INSTRUCTIONS | 9 |
| 1.5.1 Timers | 9 |
| 1.5.2 Timer accuracy | 9 |
| 1.5.3 Software errors | 9 |
| 1.5.4 Hardware errors | 10 |
| 1.5.5 Counters | 10 |
| 1.5.6 Counter formats | 10 |
| 1.5.7 High speed counter | 11 |
| 1.5.8 Sequencers | 11 |
| 1.5.9 Data handling instructions | 11 |
| 1.5.10 Data movement | 11 |
| 1.5.11 Data compression | 12 |
| 1.5.12 Arithmetic(mathematical) instructions | 12 |
| 1.5.13 Overflow | 13 |

| | |
|----------------------------------------------------|----|
| 1.5.14 Continuous control(PID) | 13 |
| 1.5.15 Master control/Master control reset(MC/MCR) | 14 |
| 1.5.16 Jump instruction | 14 |
| 1.5.17 RETURN AND | 15 |
| 1.5.18 Shift registers | 15 |

1.6 CHOSING THE CORRECT PROCESSOR 16

| | |
|----------------------------------|----|
| 1.6.1 Fault detection techniques | 17 |
| 1.6.2 Applications | 17 |
| 1.6.3 Conveyer | 17 |
| 1.6.4 System requirements | 17 |

2.PCB 22

| | |
|--------------------------|----|
| 2.1 The old days | 23 |
| 2.2 PCB packages | 23 |
| 2.3 Standarts | 24 |
| 2.4 The schematic | 24 |
| 2.5 Imperial and metric | 25 |
| 2.6 Working to grids | 26 |
| 2.7 Working from the top | 28 |
| 2.8 Tracs | 28 |
| 2.9 Pads | 31 |
| 2.10 Vias | 31 |
| 2.11 Polygons | 32 |
| 2.12 Clearances | 33 |

2.2 COMPONENTS PLACEMENT& DESIGN 34

| | |
|---------------------------|----|
| 2.2.1 Basic routing | 38 |
| 2.2.2 Finishing touches | 41 |
| 2.2.3 Single sided deign | 43 |
| 2.2.4 Double sided design | 44 |
| 2.2.5 Other layers | 44 |
| 2.2.6 Silkscreen | 44 |
| 2.2.7 Soldermask | 45 |
| 2.2.8 Mechanical layer | 46 |
| 2.2.9 Keepout | 46 |
| 2.2.10 Layer alignment | 46 |
| 2.2.11 Netlists | 47 |
| 2.2.12 Rastnest | 47 |

2.3 DESIGN RULE CHECKING 48

| | |
|-----------------------------------|----|
| 2.3.1 Forward and back annotation | 49 |
| 2.3.2 Multilayer design | 50 |
| 2.3.3 Power planes | 51 |
| 2.3.4 Good grounding | 53 |

| | |
|----------------------------------------|----|
| 2.3.5 Good bypassing | 53 |
| 2.3.6 High frequency design techniques | 54 |
| 2.3.7 Soldering | 56 |
| 2.3.8 Basic PCB manufacture | 58 |
| 2.3.9 Surface finishes | 58 |
| 2.3.10 Electrical testing | 59 |
| 2.3.11 Signature | 59 |

3. CIRCUIT ELEMENTS 60

| | |
|--------------------------|----|
| 3.1 Resistors | 60 |
| 3.2 Type of resistors | 61 |
| 3.3 Fixed resistors | 61 |
| 3.4 Carbon film resistor | 61 |
| 3.5 Metal film resistor | 63 |
| 3.6 Variable resistors | 64 |
| 3.7 CDS elements | 65 |
| 3.8 Other resistors | 66 |
| 3.9 Thermistor | 67 |
| 3.10 NTC | 67 |
| 3.11 PTC | 67 |
| 3.12 CTR | 68 |
| 3.13 Resistor color code | 69 |
| 3.14 Sensor LDR | 70 |

3.2 OPERATIONAL AMPLIFIER 73

| | |
|---------------------------------------------|----|
| 3.2.1 History | 74 |
| 3.2.2 Operation of ideal op-amps | 78 |
| 3.2.3 Applications | 79 |
| 3.2.4 Use in electronics system design | 79 |
| 3.2.5 Internal circuitry of 741 type op-amp | 82 |

3.3 DIODES 82

| | |
|------------------------------------------------|----|
| 3.3.1 Type of diodes zener diode | 83 |
| 3.3.2 Light emitting diode | 83 |
| 3.3.3 Variable capacitance diode | 83 |
| 3.3.4 Rectification/switching/regulation diode | 86 |
| 3.3.5 LED | 87 |
| 3.3.6 Schottky barrier diode | 88 |

CONCLUSION 90

REFERENCES 91

INTRODUCTION

Greenhouses evolved to be a very significant part of the agriculture and allow producing fresh fruits and vegetables all year round, also out of the season, or cultivating crops not usual for climatic conditions of certain regions. When thinking of a huge market for cut flowers, grown in the greenhouses accordingly to up-to-date fashion or by specific date, it becomes clear that protected cultivation is a very specific business area, which for successful operation needs knowledges and techniques from various sciences.

The target of the commercial purpose greenhouses, like in any other business, is to maximize profit, which depends directly on the yield grown. Very similar to human beings, plant species perform their best while being in the most comfortable environment, which in their case understands keeping the temperature, light and humidity at the optimal level for photosynthesis. The greenhouse owners make investment into the improved covering materials, able to allow more sunlight into greenhouse and yet reduce heat losses during winter. New growing media are being invented, modification and reconstruction of heating and irrigation systems is being fulfilled. Computerized environmental control systems allow an individual the ability to integrate the control of all systems involved in manipulating the growing environment, thus improving the crop development and reducing the production costs.

The first chapter represents the definition of PLC, history of PLC, types of PLC, PLC programming, hardware components of PLC systems and PLC operations

In chapter two basic knowledge about printed circuit board (PCB) and information about PCB techniques which are used in project are considered.

In chapter three general knowledge about circuit elements (resistors, NTC, PTC, OP-AMPS, diodes) which are used in project are considered.

1.INFORMATION ABOUT PLC

1.1 PLC History

PLC development began in 1968 in response to a request from an US car manufacturer (GE). The first PLCs were installed in industry in 1969.

Communications abilities began to appear in approximately 1973. They could also be used in the 70's to send and receive varying voltages to allow them to enter the analog world.

The 80's saw an attempt to: standardize communications with manufacturing automation protocol (MAP), reduce the size of the PLC, and making them software programmable through symbolic programming on personal computers instead of dedicated programming terminals or handheld programmers.

The 90's have seen a gradual reduction in the introduction of new protocols, and the modernization of the physical layers of some of the more popular protocols that survived the 1980's.

The latest standard "IEC 1131-3" has tried to merge plc programming languages under one international standard. We now have PLCs that are programmable in function block diagrams, instruction lists, C and structured text all at the same time.

1.2 HARDWARE COMPONENTS OF A PLC SYSTEM

Processor unit (CPU), Memory, Input/Output, Power supply unit, Programming device, and other devices.

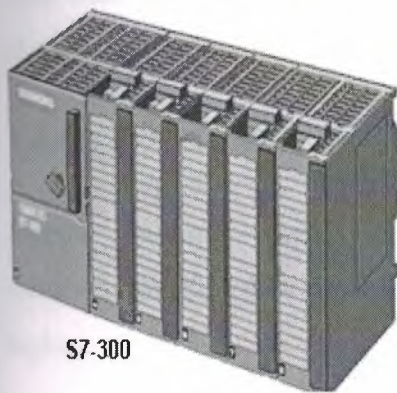


fig:1.1

1.3 CENTRAL PROCESSING UNIT (CPU)

CPU - Microprocessor based, may allow arithmetic operations, logic operators, block memory moves, computer interface, local area network, functions, etc. CPU makes a great number of check-ups of the PLC controller itself so eventual errors would be discovered early.

1.4 SYSTEM BUSESSES

The internal paths along which the digital signals flow within the PLC are called busses. The system has four busses:

- _The cpu uses the data bus for sending data between the different elements.
- _The adress bus to send the adresses of locations for accessing stored data.
- _The control bus for signals relating to internal control actions.

1.5 Memory

System (ROM) to give permanent storage for the operating system and the fixed data used by the CPU. RAM for data. This is where information is stored on the status of input and output devices and the values of timers and counters and other internal devices. EPROM for ROM's that can be programmed and then the program made permanent.

1.6 I/Q SECTIONS

Inputs monitor field devices, such as switches and sensors. Outputs control other devices, such as motors, pumps, solenoid valves, and lights.

1.7 POWER SUPPLY

Most PLC controllers work either at 24 VDC or 220 VAC. Some PLC controllers have electrical supply as a separate module, while small and medium series already contain the supply module.

1.8 PROGRAMMING DEVICE

The programming device is used to enter the required program into the memory of the processor. The program is developed in the programming device and then transferred to the memory unit of the PLC.

1.2 PLC OPERATIONS

1.2.1 INPUT RELAYS

These are connected to the outside world. They physically exist and receive signals from switches, sensors, etc... Typically they are not relays but rather they are transistors.

1.2.2 INTERNAL UTILITY RELAYS

These do not receive signals from the outside world nor do they physically exist. They are simulated relays and are what enables a PLC to eliminate external relays. There are also some special relays that are dedicated to performing only one task.

1.2.3 Counters

These do not physically exist they are simulated counters and they can be programmed to count pulses. Typically these counters can count up, down or both up and down. Since they are simulated they are limited in their counting speed. Some manufacturers also include highspeed counters that are hardware based..

1.2.4 Timers

These also do not physically exist. They come in many varieties and increments. The most common type is an on-delay type. Others include off-delay and both retentive and non-retentive types. Increments vary from 1ms through 1s.

1.2.5 .OUTPUT RELAYS

These are connected to the outside world. They physically exist and send on/off signals to solenoids, lights, etc.. They can be transistor relays or triacs depending upon the model chosen.

1.2.6 DATA STORAGE

Typically there are registers assigned to simply store data. Usually used as temporary storage for math or data manipulation. They can also typically be used to store data when power is removed from the PLC.

1.3 PLC COMMUNICATIONS

1.3.1 EXTENSION MODULES

PLC I/O number can be increased through certain additional modules by system extension through extension lines. Each module can contain extension. Extension modules can have inputs and outputs of a different nature from those on the PLC controller. When there are many I/O located considerable distances away from the PLC an economic solution is to use I/O modules and use cables to connect these, over the long distances, to the PLC.

1.3.2 REMOTE PLC'S

In some situations a number of PLCs may be linked together with a master PLC unit sending and receiving I/O data from the other units.

1.3.3 Cables

Twisted-pair cabling, often routed through steel conduit. Coaxial cable enables higher data rates to be transmitted and does not require the shielding of steel conduit.

Fiber-optic cabling has the advantage of resistance to noise, small size and flexibility.

1.3.4 Parallel communication

Parallel communication is when all the constituent bits of a word are simultaneously transmitted along parallel cables. This allows data to be transmitted over short distances at high speeds. Might be used when connecting laboratory instruments to the system.

1.3.5 PARALELL STANDARTS

The standard interface most commonly used for parallel communication is IEEE-488, and now termed as General Purpose Instrument Bus (GPIB). Parallel data communications can take place between listeners , talkers , and controllers. There are 24 lines: 8 data ,5 status control,3 handshaking and 8 ground lines.

1.3.6 Serial communication

Serial communication is when data is transmitted one bit at a time. A data word has to be separated into its constituent bits for transmission and then reassembled into the word when received. Serial communication is used for transmitting data over long distances. Might be used for the connection between a computer and a PLC.

1.4 PLC PROGRAMMING

1.4.1 PROGRAMING LANGUAGES

A program loaded into PLC systems in machine code, a sequence of binary Code numbers to represent the program instructions. Assembly language based on the use of mnemonics can be used, and a computer program called an assembler is used to translate the mnemonics into machine code.

1.4.2 PROGRAMING DEVICES

PLC can be reprogrammed through an appropriate programming device:
Programing console hand programing.

1.4.3 LADDER LOGIC

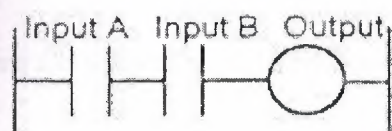
Ladder logic uses graphic symbols similar to relay schematic circuit diagrams.

Ladder diagram consists of two vertical lines representing the power rails.

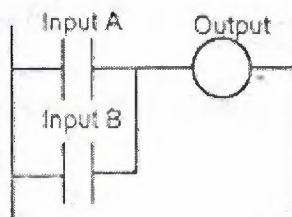
Circuits are connected as horizontal lines between these two verticals.

1.4.4 Ladder diagram features

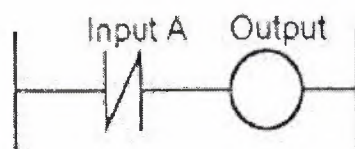
Power flows from left to right. Output on right side can not be connected directly with left side. Contact can not be placed on the right of output. Each rung contains one output at least. Each output can be used only once in the program. A particular input a/o output can appear in more than one rung of a ladder. The inputs a/o outputs are all identified by their addresses, the notation used depending on the PLC manufacturer.



AND



OR



NOT

plcmanual.com fig:1.2

1.5 PLC INSTRUCTIONS

1.5.1 Timers

Timer is an instruction that waits a set amount of time before doing something (control time). Timers count fractions of seconds or seconds using the internal CPU clock. The time duration for which a timer has been set is termed the preset and is set in multiples of the time base used.

Most manufacturers consider timers to behave like relays with coils which when energized result in the closure or opening of contacts after some preset time. The timer is thus treated as an output for a rung with control being exercised over pairs of contacts elsewhere. Others treat a timer as a delay block which when inserted in a rung delays signals in that rung reaching the output.

1.5.2 TIMER ACCURACY

There are software and Hardware Errors when using a timer.

1.5.3 SOFTWARE ERRORS

Input error depending upon when the timer input turns on during the scan cycle.

Output error depending upon when in the ladder the timer actually “times out” and when the plc finishes executing the program to get to the part of the scan when it updates the outputs. Total software errors is the sum of the input and output errors.

1.5.4 Hardware errors

There is a hardware input error as well as a hardware output error. The hardware input error is caused by the time it takes for the plc to actually realize that the input is on when it scans its inputs. Typically this duration is about 10ms. The hardware output error is caused by the time it takes from when the plc tells its output to physically turn on until the moment it actually does. Typically a transistor takes about 0.5ms whereas a mechanical relay takes about 10ms.

1.5.5 Counters

A counter is set to some preset value and, when this value of input pulses has been received it will operate its contacts. The counter accumulated value ONLY changes at the off to on transition of the pulse input. Typically counters can count from 0 to 9999, -32,768 to +32,767 or 0 to 65535. The normal counters are typically "software" counters - they don't physically exist in the plc but rather they are simulated in software. A good rule of thumb is simply to always use the normal (software) counters unless the pulses you are counting will arrive faster than 2X the scan time.

1.5.6 Counter Formats

Some manufacturers consider the counter as a relay and consist of two basic elements:

One relay coil to count input pulses and one to reset the counter, and the associated contacts of the counter being used in other rungs.

Others (Siemens for example) treat the counter as an intermediate block in a rung from which signals emanate when the count is attained.

1.5.7 High speed counter

Most manufacturers also include a limited number of high-speed counters (HSC). Typically a high-speed counter is a “hardware” device. Hardware counters are not dependent on scan time.

1.5.8 Sequencers

The sequencer is a form of counter that is used for sequential control. It replaces the mechanical drum sequencer that was used to control machines that have a stepped sequence of repeatable operations. The PLC sequencer consists of a master counter that has a range of presets counts corresponding to the different steps and so, as it progresses through the count, when each preset count is reached can be used to control outputs.

1.5.9 Data Handling Instructions

Timers, counters and individual relays are all concerned with the handling of individual bits, i.e. single on-off signal. PLC operations involve blocks of data representing a value, such blocks being termed words.

Data handling consists of operations involving moving or transferring numeric information stored in one memory word location to another word in a different location, comparing data values and carrying out simple arithmetic operations.

1.) A register is where data can be stored. Each data register can store a binary word of usually 8 or 16 bits. The number of bits determines the size of the number that can be stored (2ⁿ)

2.) 4-bit register can store a positive number between 0 and +15. 8-bit 0 and +255. 16-bit: 0 and +65535.

1.5.10 Data movement instructions

There are typically 2 common instruction sets. The single instruction is commonly called MOV (move) copies a value from one address to another. The MOV instruction needs to know 2 things: Source - where the data we want to move is located. Destination - the location where the data will be moved to. We write an address here. Also, the data can be moved to the physical outputs.

1.5.11 Data comparison

The data comparison instruction gets the PLC to compare two data values.

Thus it might be to compare a digital value read from some input device with a second value contained in a register. PLCs generally can make comparisons for:

Less than(< or LESS), equal to(= or EQU), less than or equal to(<= or LEQ), Greater than(> or GRT), greater than or equal to(>= or GEQ) and not equal to(NEQ)

1.5.12 Arithmetic (mathematical) Instructions

PLCs almost always include math functions to carry out some arithmetic operations:

Addition (ADD) - The capability to add one piece of data to another.

Subtraction (SUB) - The capability to subtract one piece of data from another.

Multiplication (MUL) - The capability to multiply one piece of data by another.

Division (DIV) - The capability to divide one piece of data from another.

15.13 Overflow

Typically the memory locations are 16-bit locations. If a result is greater than the value that could be stored in a memory location then we get an overflow. The plc turns on an internal relay that tells us an overflow has happened. We get an overflow if the number is greater than 65535 ($2^{16}=65536$).

Depending on the plc, we would have different data in the destination location. Some use 32-bit math which solves the problem. If we're doing division, and we divide by zero the overflow bit turns on.

15.14 Continuous control (PID Instruction)

Continuous control of some variable can be achieved by comparing the actual value of the variable with the desired set value and then giving an output depending on the control law required. Many PLCs provide the PID calculation to determine the controller output as a standard routine. All that is then necessary is to pass the desired parameters, i.e. the values of Kp, Ki, and KD, and input/output locations to the routine via the PLC program.

Control instructions are used to enable or disable a block of logic program or to move execution of a program from one place to another place.

The control instructions include:

- MasterControl instruction (MC/MCR)
- Jump to label instruction (JMP)
- Label instruction (LBL)
- Jump to Subroutine instruction (JSR)
- Subroutine instruction (SBR)
- Return from Subroutine instruction (RET)
- Shift Registers

1.5.15 Master Control/ Master Control Reset (MC/MCR)

When large numbers of outputs have to be controlled, it is sometimes necessary for whole sections of program to be turned on or off when certain criteria are realized. This could be achieved by including a MCR instruction. A MCR instruction is an output instruction.

When the rung with the first MCR instruction is true, the first MCR instruction is high and the outputs of the rung in the controlled zone can be energized or dennergized according to their rung conditions. When the this rung is false, all the outputs in the zone are dennergized, regardless their rung conditions.

Timers should not be used inside the MC/MCR block because some manufacturers will reset them to zero when the block is false whereas other manufacturers will have them retain the current time state. Counters typically retain their current counted value.

1.5.16 Jump Instructions

The JUMP instructions allow to break the rung sequence and move the program execution from one rung to another or to a subroutine. The Jump is a controlled output. You can jump forward or backward. You can use multiple jump to the same label. Jumps within jumps are possible there are:

1. Jump to Label.
2. Jump to subroutine

1.5.17 RETURN AND

A Return from Subroutine instruction marks the end of Subroutine instruction. When the rung condition of this instruction is true, it causes the PLC to resume execution in the calling program file at the rung following the Jump to Subroutine instruction in the calling program.

When a Return from Subroutine instruction is not programmed in a subroutine file, the END instruction automatically causes the PLC to move execution back to the rung following the Jump to Subroutine instruction. A Jump to Subroutine instruction can be used either in a main application program or a subroutine program to call another subroutine program.

1.5.18 Shift Registers

The shift register is a number of internal relays grouped together (normally 8, 16, or 32) which allow stored bits to be shifted from one relay to another. The grouping together of internal relays to form a shift register is done automatically by a PLC when the shift register function is selected. This is done by using the programming code against the internal relay number that is to be the first in the register array.

Shift registers can be used where a sequence of operations is required or to keep track of particular items in a production system. The shift register is most commonly used in conveyor systems, labeling or bottling applications, etc.

1.6 CHOOSING THE CORRECT PROCESSOR

For Selecting Modular Processors the following Criteria examined include: I/O points (local I/O points and expandable points). Each PLC processor will only be capable of working with a limited number of each type of I/O modules.

Memory size (for data storage or program storage) and Performance (scan time) depends on the processor. The size of program is dependent upon the complexity of the control problem and the skill and style of the programmer

The required operating speed for all the I/O must be determined, with a PLC selected to match. This requires the estimation of the program size and the proportion of slow instructions. The scan speed is normally expressed in terms of ms/K for a stated mix of simple and complex instructions. A PLC with an appropriate memory capacity and speed can be selected.

For any particular application it is essential to ensure that the PLC selected can handle the required operations. When a communications facility is required we need to determine whether the built-in port is adequate for the application, or whether a separate module will be required.

1.6.1 Fault detection techniques

For any PLC controlled plant, by far the greater percentage of the faults are likely to be with sensors, actuators, and wiring rather than with PLC itself. The faults within the PLC most are likely to be in the input/output channels or power supply than in the CPU.

1.6.2 Applications

1.6.3 Conveyor

This simple application is for a conveyor (moving material machine) and how we implement it using ladder diagram and instruction list.



1.6.4 SYSTEM REQUIREMENTS

1. A plc is used to start and stop the motors of a segmented conveyor belt, this allows only belt sections carrying a copper plate to move.

2. The system have three segmented conveyor belts, each segment runs by a motor.

3. A proximity switch located at the end of each segment to detect the position of the plate.

4. The first conveyor segment is always on.

5. The second conveyor segment turns on when the proximity switch in the first segment detects the plate.

6. when the proximity switch at the second conveyor detects the plate, the third segment conveyor turns ON.

7. the second conveyor is stopped, when the plate is out of detection range of the second proximity switch, after 20 seconds.

8. the third conveyor is stopped after 20 seconds, when the proximity switch located at the segment doesn't detect the plate.

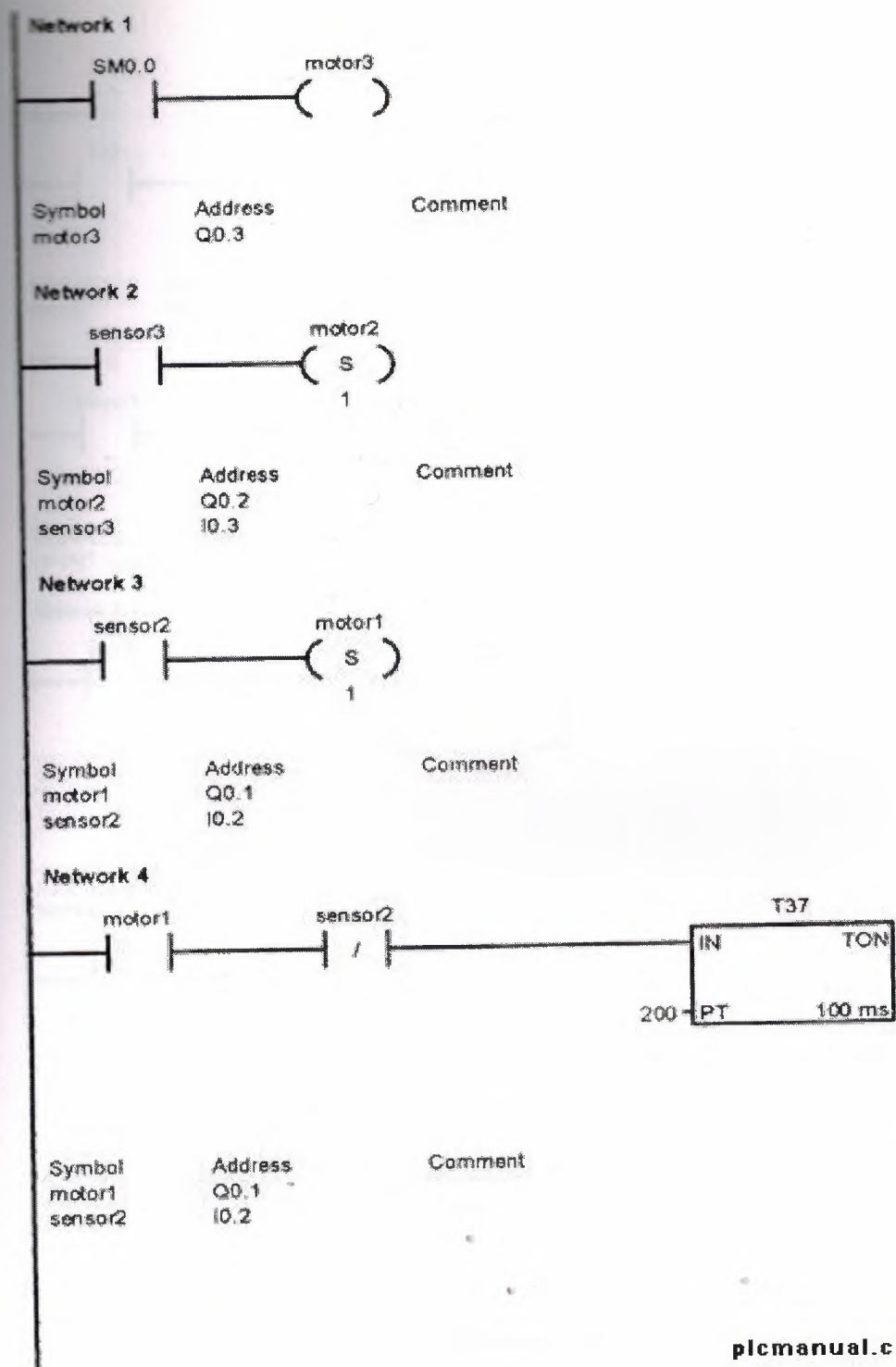


Fig:1.3

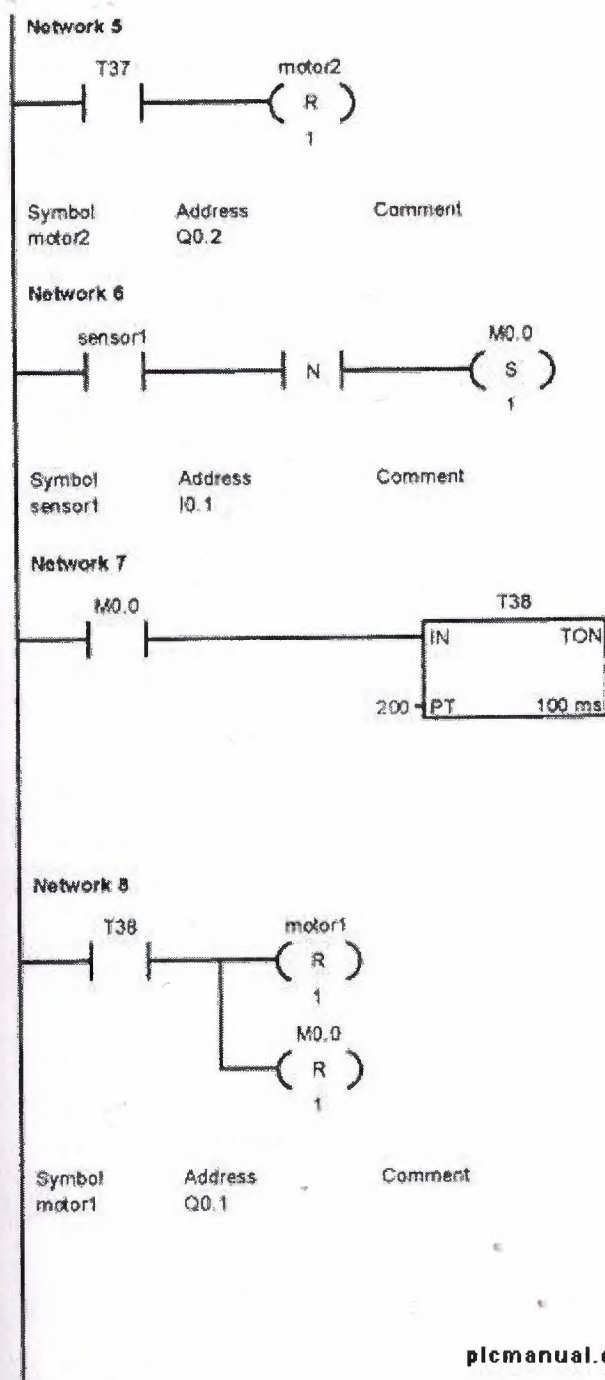


Fig:1.4

Network 1

LD SM0.0
= motor3

| Symbol | Address |
|--------|---------|
| motor3 | Q0.3 |

Network 2

LD sensor3
S motor2, 1

| Symbol | Address |
|---------|---------|
| motor2 | Q0.2 |
| sensor3 | I0.3 |

Network 3

LD sensor2
S motor1, 1

| Symbol | Address |
|---------|---------|
| motor1 | Q0.1 |
| sensor2 | I0.2 |

Network 4

LD motor1
AN sensor2
TON T37, 200

| Symbol | Address |
|---------|---------|
| motor1 | Q0.1 |
| sensor2 | I0.2 |

Network 5

LD T37
R motor2, 1

| Symbol | Address |
|--------|---------|
| motor2 | Q0.2 |

Network 6

LD sensor1
ED
S M0.0, 1

| Symbol | Address |
|---------|---------|
| sensor1 | I0.1 |

Network 7

LD M0.0
TON T38, 200

Network 8

LD T38
R motor1, 1
R M0.0, 1

| Symbol | Address |
|--------|---------|
| motor1 | Q0.1 |

plcmanual.com

2. PCB

You've designed your circuit, perhaps even bread boarded a working prototype, and now it's time to turn it into a nice Printed Circuit Board (PCB) design. For some designers, the PCB design will be a natural and easy extension of the design process. But for many others the process of designing and laying out a PCB can be a very daunting task. There are even very experienced circuit designers who know very little about PCB design,

and as such leave it up to the "expert" specialist PCB designers. Many companies even have their own dedicated PCB design departments. This is not surprising, considering that it often takes a great deal of knowledge and talent to position hundreds of components and thousands of tracks into an intricate (some say artistic) design that meets a whole host of physical and electrical requirements. Proper PCB design is very often an integral part of a design. In many designs (high speed digital, low level analog and RF to name a few) the PCB layout may make or break the operation and electrical performance of the design. It must be remembered that PCB traces have resistance, inductance, and capacitance, just like your circuit does. This article is presented to hopefully take some of the mystery out of PCB design. It gives some advice and

"rules of thumb" on how to design and lay out your PCBs in a professional manner. It is, however, quite difficult to try and "teach" PCB design. There are many basic rules and good practices to follow, but apart from that PCB design is a highly creative and individual process. It is like trying to teach someone how to paint a picture. Everyone will have their own unique style, while some people may have no creative flair at all! Indeed, many PCB designers like to think of PCB layouts as works of art, to be admired for their beauty and elegance. "If it looks good, it'll work good." is an old catch phrase.

2.1 The Old Days

Back in the pre-computer CAD days, PCBs were designed and laid out by hand using adhesive tapes and pads on clear drafting film. Many hours were spent slouched over a fluorescent light box, cutting, placing, ripping up, and routing tracks by hand. Bishop Graphics, Letraset, and even Dalo pens will be names that evoke fond, or not

so fond memories. Those days are well and truly gone, with computer based PCB design having replaced this method completely in both hobbyist and professional electronics. Computer based CAD programs allow the utmost in flexibility in board design and editing over the traditional techniques. What used to take hours can now be done in seconds.

2.2 PCB Packages

There are many PCB design packages available on the market, a few of which are freeware, shareware, or limited component full versions. Protel is the defacto industry standard package in Australia. Professionals use the expensive high end Windows based packages such as 99SE and DXP. Hobbyists use the excellent freeware DOS based Protel AutoTrax program, which was, once upon a time, the high-end package of choice in Australia. Confusingly, there is now another Windows based package also called AutoTrax EDA. This is in no way related to the Protel software. This article does not focus on the use of any one package, so the information can be applied to almost any PCB package available. There is however, one distinct exception. Using a PCB only package, which does not have schematic capability, greatly limits what you can do with the package in the professional sense. Many of the more advanced techniques to be described later require access to a compatible schematic editor program. This will be explained when required

2.3 Standards

There are industry standards for almost every aspect of PCB design. These standards are controlled by the former Institute for Interconnecting and Packaging Electronic Circuits, who are now known simply as the IPC(www.ipc.org). There is an IPC standard for every aspect of PCB design, manufacture, testing, and anything else that you could ever need.

2.4 The Schematic

Before you even begin to lay out your PCB, you **MUST** have a complete and accurate schematic diagram. Many people jump straight into the PCB design with nothing more than the circuit in their head, or the schematic drawn on loose post-it notes with no pin numbers and no order. This just isn't good enough, if you don't have an

accurate schematic then your PCB will most likely end up a mess, and take you twice as long as it should. "Garbage-in, garbage-out" is an often used quote, and it can apply equally well to PCB design. A PCB design is a manufactured version of your schematic, so it is natural for the PCB design to be influenced by the original schematic. If your schematic is neat, logical and clearly laid out, then it really does make your PCB design job a lot easier. Good practice will have signals flowing from inputs at the left to outputs on the right. With electrically important sections drawn correctly, the way the designer would like them to be laid out on the PCB. Like putting bypass capacitors next to the component they are meant for. Little notes on the schematic that aid in the layout

are very useful. For instance, "this pin requires a guard track to signal ground", makes it clear to the person

laying out the board what precautions must be taken. Even if it is you who designed the circuit and drew the schematic, notes not only remind yourself when it comes to laying out the board, but they are useful for people reviewing the design. Your schematic really should be drawn with the PCB design in mind. It is outside the scope of this article to go into details on good schematic design, as it would require a complete article in its own right.

2.5 Imperial and Metric

The first thing to know about PCB design is what measurement units are used and their common terminologies, as they can be awfully confusing!

As any long time PCB designer will tell you, you should always use imperial units (i.e. inches) when designing PCBs. This isn't just for the sake of nostalgia, although that is a major reason! The majority of electronic components were (and still are) manufactured with imperial pin spacing. So this is no time to get stubborn and refuse to use anything but metric units, metric will make laying out of your board a lot harder and a lot messier. If you are young enough to have been raised in the metric age then you had better start learning what inches are all about and how to convert them. An old saying for PCB design is "thou shall use thous". A tad confusing until you know what a "thou" is.

A "thou" is 1/1000th of an inch, and is universally used and recognised by PCB designers and manufacturers everywhere. So start practicing speaking in terms of "10 thou spacing" and "25 thou grid", you'll sound like a professional in no time! Now that you understand what a thou is, we'll throw another spanner in the works with the term "mil" (or "mils").

"mil" is the same as 1 thou, and is NOT to be confused with the millimeter (mm), which is often spoken the same as "mil". The term "mil" comes from 1 thou being equal to 1 mili inch. As a general rule avoid the use of "mil" and stick to "thou", it's less confusing when trying to explain PCB dimensions to those metricated non-PCB

people. Some PCB designers will tell you not to use metric millimeters for ANYTHING to do with a PCB design. In the practical world though, you'll have to use both imperial inches (thous) and the metric millimeter (mm). So which units do you use for what? As a general rule, use thous for tracks, pads, spacings and grids, which are most of your basic "design and layout" requirements. Only use mm for "mechanical and manufacturing" type requirements like hole sizes and board dimensions. You will find that many PCB manufacturers will follow these basic guidelines also, for when they ask

you to provide details for a quote to manufacture your board. Most manufacturers use metric size drills, so specifying imperial size holes really is counterproductive and can be prone to errors. Just to confuse the issue even further, there are many components (new surface mount)

which have metric pin spacing and dimensions. So you'll often have to design some component footprints using metric grids and pads. Many component datasheets will also have metric dimensions even though the spacing are designed to an imperial grid. If you see a "weird" metric dimension like 1.27mm in a component, you can be

pretty sure it actually has a nice round imperial equivalent. In this case 1.27mm is 50 thou. Yes, PCB design can be confusing! So whatever it is you have to do in PCB design you'll need to become an expert at imperial to metric conversion, and vice-versa. To make your life easier though, all the major PCB drafting packages have a single "hot key" to convert between imperial and metric units instantly ("Q" on Protel for instance). It will help you greatly if you memorise a few key conversions, like **100 thou (0.1 inch) = 2.54mm**, and **200 thou (0.2 inch) = 5.08mm** etc Values of 100 thou and above are very often expressed in inches instead of thous. So 0.2 inch is more commonly used than 200thou. 1 inch is also commonly known as 1 "pitch". So it is common to hear the phrase "0.1 inch pitch", or more simply "0.1 pitch" with the inches units being assumed. This is often used for pin spacing on components. 100 thou is a basic "reference point" for all aspects of PCB design, and a vast array of common component lead spacing are multiples or fractions of this basic unit. 50 and 200 thou are the most common. Along with the rest of the world, the IPC standards have all been metricated, and only occasionally refer to imperial units. This hasn't really converted the PCB industry though. Old habits die hard, and imperial still reigns supreme in many areas of practical usage.

2.6 Working to Grids

The second major rule of PCB design, and the one most often missed by beginners, is to lay out your board on a fixed grid. This is called a "snap grid", as your cursor, components and tracks will "snap" into fixed grid positions.

Not just any size grid mind you, but a fairly coarse one. 100 thou is a standard placement grid for very basic

through hole work, with 50 thou being a standard for general tracking work, like running tracks between throughhole pads. For even finer work you may use a 25 thou snap grid or even lower. Many designers will argue over the merits of a 20 thou grid vs a 25 thou grid for instance. In practice, 25 thou is often more useful as it allows you to go exactly half way between 50 thou spaced pads. Why is a coarse snap grid so important? It's important because it will keep your components neat and symmetrical; aesthetically pleasing if you may. It's not just for aesthetics though - it makes future editing,

dragging, movement and alignment of your tracks, components and blocks of components easier as your layout grows in size and complexity. A bad and amateurish PCB design is instantly recognisable, as many of the tracks will not line up exactly in the

center of pads. Little bits of tracks will be "tacked" on to fill in gaps etc. This is the result of not using a snap grid effectively. Good PCB layout practice would involve you starting out with a coarse grid like 50 thou and using a progressively finer snap grid if your design becomes "tight" on space. Drop to 25 thou and 10 thou for finer routing and placement when needed. This will do 99% of boards. Make sure the finer grid you choose is a nice even division of your standard 100 thou. This means 50, 25, 20, 10, or 5 thou. Don't use anything else, you'll regret it. A good PCB package will have hotkeys or programmable macro keys to help you switch between different snap grid sizes instantly, as you will need to do this often. There are two types of grids in a PCB drafting package, a snap grid as discussed, and a "visible" grid. The visible grid is an optional on-screen grid of solid or dashed lines, or dots. This is displayed as a background behind your design and helps you greatly in lining up components and tracks. You can have the snap grid and visible grid set to different units (metric or imperial), and this is often very helpful. Many designers prefer a 100 thou visible grid and rarely vary from that.

Some programs also have what is called an "Electrical" grid. This grid is not visible, but it makes your cursor "snap" onto the center of electrical objects like tracks and pads, when your cursor gets close enough. This is extremely useful

for manual routing, editing and moving objects. One last type of grid is the “Component” grid. This works the same as the snap grid, but it’s for component movement only. This allows you to align components up to a different grid. Make sure you make it a multiple of your Snap grid. When you start laying out your first board, snap grids can feel a bit “funny”, with your cursor only being able to be moved in steps. Unlike normal paint type packages which everyone is familiar with. But it’s easy to get used to, and your PCB designs will be one step closer to being neat and professional.

2.7 Working from the top

PCB design is always done looking from the top of your board, looking through the various layers as if they were transparent. This is how all the PCB packages work. The only time you will look at your board from the bottom is for manufacturing or checking purposes. This “through the board” method means that you will have to get used to reading text on the bottom layers as a mirror image, get used to it!

2.8 Tracks

There is no recommended standard for track sizes. What size track you use will depend upon (in order of importance) the electrical requirements of the design, the routing space and clearance you have available, and your own personal preference. Every design will have a different set of electrical requirements which can vary between tracks on the board. All but basic non-critical designs will require a mixture of track sizes. As a general rule though, the bigger the track width, the better. Bigger tracks have lower DC resistance, lower inductance, can be easier and cheaper for the manufacturer to etch, and are easier to inspect and rework. The lower limit of your track width will depend upon the “track/space” resolution that your PCB manufacturer is capable of. For example, a manufacturer may quote a 10/8 track/space figure. This means that tracks can be no less than 10 thou wide, and the spacing between tracks (or pads, or any part of the copper) can be no less than 8 thou. The figures are almost always quoted in

thou's, with track width first and then spacing. Real world typical figures are 10/10 and 8/8 for basic boards. The IPC standard recommends 4thou as being a lower limit. Once you get to 6thou tracks and below though, you are getting into the serious end of the business, and you should be consulting your board manufacturer first. The lower the track/space figure, the greater care the manufacturer has to take when aligning and etching the board. They will pass this cost onto you, so make

sure that you don't go any lower than you need to. As a guide, with "home made" PCB manufacturing processes like laser printed transparencies and pre-coated photo resist boards, it is possible to easily get 10/10 and even 8/8 spacing. Just because a manufacturer can achieve a certain track/spacing, it is no reason to "push the limits" with your design. Use as big a track/spacing as possible unless your design parameters call for something smaller. As a start, you may like to use say 25 thou for signal tracks, 50 thou for power and ground tracks, and 10-15 thou for going between IC and component pads. Some designers though like the "look" of smaller signal tracks

like 10 or 15 thou, while others like all of their tracks to be big and "chunky". Good design practice is to keep tracks as big as possible, and then to change to a thinner track only when required to meet clearance requirements. Changing your track from large to small and then back to large again is known as "necking", or "necking down". This is often required when you have to go between IC or component pads. This allows you to have nice big low impedance tracks, but stil have the flexibility to route between tight spots.

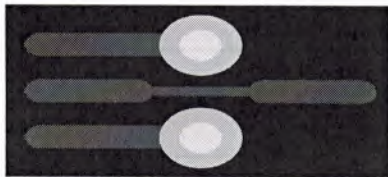


fig:2.1

In practice, your track width will be dictated by the current flowing through it, and the maximum temperature rise of the track you are willing to tolerate. Remember that every track will have a certain amount of resistance, so the

track will dissipate heat just like a resistor. The wider the track the lower the resistance. The thickness of the

copper on your PCB will also play a part, as will any solder coating finish.

The thickness of the copper on the PCB is nominally specified in ounces per square foot, with 1oz copper being the most common. You can order other thicknesses like 0.5oz, 2oz and 4oz. The thicker copper layers are useful for high current, high reliability designs.

The calculations to figure out a required track width based on the current and the maximum temperature rise are a little complex. They can also be quite inaccurate, as the standard is based on a set of non-linear graphs based on measured data from around half a century ago. These are still reproduced in the IPC standard.

A handy track width calculator program can be found at www.ultracad.com/calc.htm, and gives results based on the IPC graphs. As a rule of thumb, a 10degC temperature rise in your track is a nice safe limit to design around. A handy reference table has been included in this article to give you a list of track widths vs current for a 10degC rise. The DC resistance in milli ohms per inch is also shown. Of course, the bigger the track the better, so don't just blindly stick to the table.

| Track Width Reference Table (for 10deg C temp rise). Track Width is in Thous (mils) | | | |
|-------------------------------------------------------------------------------------|---------------|----------------|-----------------|
| Current (Amps) | Width for 1oz | Width for 2 oz | milli Ohms/Inch |
| 1 | 10 | 5 | 52 |
| 2 | 30 | 15 | 17.2 |
| 3 | 50 | 25 | 10.3 |
| 4 | 80 | 40 | 6.4 |
| 5 | 110 | 55 | 4.7 |
| 6 | 150 | 75 | 3.4 |
| 7 | 180 | 90 | 2.9 |
| 8 | 220 | 110 | 2.3 |
| 9 | 260 | 130 | 2.0 |
| 10 | 300 | 150 | 1.7 |

Table:2.1

2.9 Pads

Pad sizes, shapes and dimensions will depend not only upon the component you are using, but also the manufacturing process used to assemble the board, among other things. There are a whole slew of standards and theories behind pad sizes and layouts, and this will be explained later. Suffice it to say at this stage that your PCB package should come with a set of basic component libraries that will get you started. For all but the simplest boards though, you'll have to modify these basic components to suit your purpose. Over time you will build up your own library of components suitable for various requirements.

There is an important parameter known as the pad/hole ratio. This is the ratio of the pad size to the hole size.

Each manufacturer will have their own minimum specification for this. As a simple rule of thumb, the pad should be at least 1.8 times the diameter of the hole, or at least 0.5mm larger. This is to allow for alignment tolerances on the drill and the artwork on top and bottom layers. This ratio gets more important the smaller the pad and hole

become, and is particularly relevant to vias.

There are some common practices used when it comes to generic component pads. Pads for leaded components like resistors, capacitors and diodes should be round, with around 70 thou diameter being common.

Dual In Line (DIL) components like IC's are better suited with oval shaped pads (60 thou high by 90-100 thou wide is common). Pin 1 of the chip should always be a different pad shape, usually rectangular, and with the same dimensions as the other pins.

Most surface mount components use rectangular pads, although surface mount SO package ICs should use oval pads. Again, with pin 1 being rectangular.

Other components that rely on pin numbering, like connectors and SIP resistor packs, should also follow the "rectangular pin 1" rule.

Octagonal pads are seldom used, and should generally be avoided.

As a general rule, use circular or oval pads unless you need to use rectangular.

2.10 Vias

Vias connect the tracks from one side of your board to another, by way of a hole in your board. On all but cheap home made and low end commercial prototypes, vias are made with electrically plated holes, called Plated Through Holes (PTH). Plated through holes allow electrical connection between different layers on your board.

What is the difference between a via and a pad? Practically speaking there is no real difference, they are both just electrically plated holes. But there are differences when it comes to PCB design packages. Pads and Vias are, and should be, treated differently. You can globally edit them separately, and do some more advanced things to be discussed later. So don't use a pad in place of a via, and vice-versa.

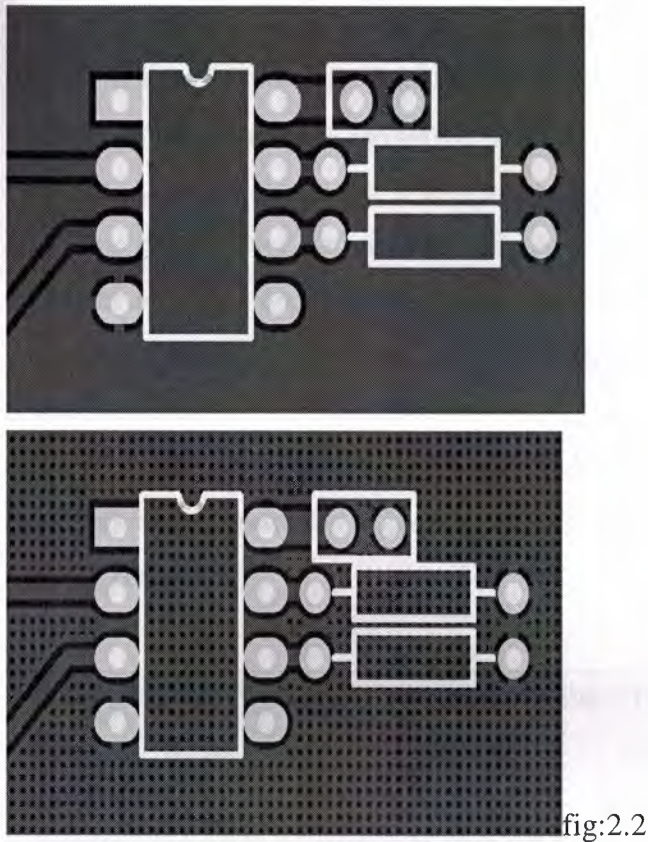
Holes in vias are usually a fair bit smaller than component pads, with 0.5-0.7mm being typical.

Using a via to connect two layers is commonly called "stitching", as you are effectively electrically stitching both layers together, like threading a needle back and forth through material. Throw the term stitching a few times into a conversation and you'll really sound like a PCB professional!

2.11 Polygons

"Polygons" are available on many PCB packages. A polygon automatically fills in (or "floods") a desired area with copper, which "flows" around other pads and tracks. They are very useful for laying down ground planes. Make sure you place polygons after you have placed all of your tracks and pads.

Polygon can either be "solid" fills of copper, or "hatched" copper tracks in a crisscross fashion. Solid fills are preferred, hatched fills are basically a thing of the past.



An example of a “Solid Polygon Fill” (top), and a “Hatched Polygon Fill” (bottom)

2.12 Clearances

Electrical clearances are an important requirement for all boards. Too tight a clearance between tracks and pads may lead to “hairline” shorts and other etching problems during the manufacturing process. These can be very hard to fault find once your board is assembled. Once again, don’t “push the limits” of your manufacturer unless

you have to, stay above their recommended minimum spacing if at all possible. At least 15 thou is a good clearance limit for basic through hole designs, with 10 thou or 8 thou being used for more dense surface mount layouts. If you go

below this, it's a good idea to consult with your PCB manufacturer first. For 240V mains on PCB's there are various legal requirements, and you'll need to consult the relevant standards

if you are doing this sort of work. As a rule of thumb, an absolute minimum of 8mm (315 thou) spacing should be allowed between 240V tracks and isolated signal tracks. Good design practice would dictate that you would have much larger clearances than this anyway.

For non-mains voltages, the IPC standard has a set of tables that define the clearance required for various voltages. A simplified table is shown here. The clearance will vary depending on whether the tracks are on an internal layers or the external surface. They also vary with the operational height of the board above sea level,

due to the thinning of the atmosphere at high altitudes. Conformal coating also improves these figures for a given clearance, and this is often used on military spec PCBs.

| Clearances for Electrical Conductors | | | |
|---------------------------------------------|-----------------|-----------------------------|-----------------------------|
| Voltage (DC or Peak AC) | Internal | External (<3050m) | External (>3050m) |
| 0-15V | 0.05mm | 0.1mm | 0.1mm |
| 16-30V | 0.05mm | 0.1mm | 0.1mm |
| 31-50V | 0.1mm | 0.6mm | 0.6mm |
| 51-100V | 0.1mm | 0.6mm | 1.5mm |
| 101-150V | 0.2mm | 0.6mm | 3.2mm |
| 151-170V | 0.2mm | 1.25mm | 3.2mm |
| 171-250V | 0.2mm | 1.25mm | 6.4mm |
| 251-300V | 0.2mm | 1.25mm | 12.5mm |
| 301-500V | 0.25mm | 2.5mm | 12.5mm |

Table:2.2

2.2 COMPONENTS PLACEMENT & DESIGN

An old saying is that PCB design is 90% placement and 10% routing. Whilst the actual figures are of no importance, the concept that component placement

is by far the most important aspect of laying out a board certainly holds true. Good component placement will make your layout job easier and give the best electrical performance. Bad component placement can turn your routing job into a nightmare and give poor electrical

performance. It may even make your board unmanufacturable. So there is a lot to think about when placing components!

Every designer will have their own method of placing components, and if you gave the same circuit (no matter how simple) to 100 different experienced designers you'd get a 100 different PCB layouts every time. So there is no absolute right way to place your components. But there are quite a few basic rules which will help ease your routing, give you the best electrical performance, and simplify large and complex designs.

At this point it is a good idea to give you an idea of the basic steps required to go about laying out a complete board:

- . Set your snap grid, visible grid, and default track/pad sizes.
- . Throw down all the components onto the board.
- . Divide and place your components into functional "building blocks" where possible.
- . Identify layout critical tracks on your circuit and route them first.
- . Place and route each building block separately, off the board.
- . Move completed building blocks into position on your main board.
- . Route the remaining signal and power connections between blocks.
- . Do a general "tidy up" of the board.
- . Do a Design Rule Check.
- ☐ Get someone to check it

This is by no means a be-all and end-all check list, it's highly variable depending on many factors. But it is a good general guide to producing a professional first-class layout. Lets look in more detail at the procedure described above.

We have already looked at the grids and track/pad sizes, these should be the first things that you set up before you start doing anything. No exceptions!

Many people like to jump straight into placing all the components into what they think is the most optimum position on the board, all in one hit. Whilst this can work for small circuits, you don't have much of a hope when you have more complex circuits with hundreds of components spread across many functional circuit blocks.

Why?, because it's very easy to run out of "routing space", which is the room to lay down all your tracks. If you fix all your component positions and then try to route everything, you can easily paint yourself into a corner so to speak. Alternatively, if you space the components out too much, you can end up with a large board that does not make efficient use of space. The hallmark of an inexperienced designer is a board that has every component evenly spaced out, and then has thousands of tracks and vias crisscrossing the board. It might work, but it can be ugly and inefficient, not to mention bigger and more expensive to manufacture.

The best way to start your layout is to get ALL of your components onto the screen first.

If you have a companion schematic package, then the simplest way to do this is to get your PCB program to import your schematic design and select all the components automatically. This will also be discussed later. If all you have is a PCB program, then you'll have to select each component from the library and place them down manually.

With all the components on screen, you should get a good indication of whether or not your parts will easily fit onto the size (and shape) of board that you require. If it looks like it's going to be a tight fit then you know that you will have to work hard to try and keep the component spacing "tight", and the tracking as efficient as possible. If it looks like you have plenty of room then you can be a bit more liberal in your layout. Of course, if it looks like you have buckleys chance of getting your components on the board, you'll have to go back to the drawing board. Now analyse your schematic and determine which parts of the design can be broken up into "building blocks". Often this is fairly obvious. Say for example you have a complex looking active filter in your circuit. This would typically have a single input line and a single output line, but it will have lots of components and connections as part of the filter. This is

a classic “building block” circuit, and one that lends itself well to combining all of these parts together in the same location. So you would grab all of these parts and start to rearrange them into their own little layout off to one side of your board. Don’t worry too much about where the actual block goes on your board yet. You will also need to partition off electrically sensitive parts of your design into bigger blocks. One major example is with mixed digital and analog circuits. Digital and analog just do not mix, and will need to be physically and electrically separated. Another example is with high frequency and high current circuits, they do not mix with low frequency and low current sensitive circuits. More about this later. As a general rule, your components should be neatly lined up. Having ICs in the same direction, resistors in neat columns, polarised capacitors all around the same way, and connectors on the edge of the board. Don’t do this at the expense of having an electrically poor layout, or an overly big board though. Electrical parameters should always take precedence over nicely lined up components. Symmetry is really nice in PCB design, it’s aesthetically pleasing and just “looks right”. If you have something like two identical building block circuits side by side, and one is laid out slightly differently, it sticks out like a sore thumb. If you have placed your components wisely, 90% of your work will be done. The last 10% should just be joining the dots so to speak. Well, not quite, but good placement is a good majority of your work done. Once you are happy with the component placements, you can start to route all the different building blocks separately. When finished, it is then often a simple matter to move and arrange the building blocks into the rest of your design.

The Design Rule Check (DRC) will be covered later, but it is an essential step to ensuring that your board is correct before manufacture. A DRC basically checks for correct connectivity of your tracks, and for correct widths and clearances. Getting someone to check your board may sound like an overly bureaucratic process, but it really is a vital step. No matter how experienced you are at PCB design, there will always be something you overlooked. A fresh pair of eyes and a different mindset will pick up problems you would never see. If you don’t have anyone to check your board over and don’t have DRC capability, then you’ll have to do it yourself. Get a printout of your schematic

and highlighter pen. Now, compare every single electrical “net” connection on your board with the schematic, net by net. Highlight each net on the schematic as you complete it. When you are finished, there should be no electrical connections left that aren’t highlighted. You can now be fairly confident that your board is electrically correct.

2.2.1 Basic Routing

Now it’s time for some basic routing rules. Routing is also known as “tracking”.

Routing is the process of laying down tracks to connect components on your board. An electrical connection between two or more pads is known as a “net”. Keep nets as short as possible. The longer your total track length, the greater it’s resistance, capacitance and inductance. All of which can be undesirable factors.

Tracks should only have angles of 45 degrees. Avoid the use of right angles, and under no circumstances use an angle greater than 90 degrees. This is important to give a professional and neat appearance to your board. PCB packages will have a mode to enforce 45 degree movements, make use of it. There should never be a need to turn it off. Contrary to popular belief, sharp right angle corners on tracks don’t produce measurable EMI or other problems. The reasons to avoid right angles are much simpler - it just doesn’t look good, and it may have some manufacturing implications.

Forget nice rounded track corners, they are harder and slower to place and have no real advantage. Stick to 45 degree increments. Rounded track bends belong to the pre-CAD taped artwork era.

“Snake” your tracks around the board, don’t just go “point to point”. Point to point tracking may look more efficient to a beginner at first, but there are a few reasons you shouldn’t use it. The first is that it’s ugly, always an important factor in PCB design! The second is that it is not very space efficient when you want to run more tracks on other layers.

Enable your Electrical grid, which is sometimes referred to as a “snap to center” or “snap to nearest” option. Let the software find the centers of pads and ends of tracks automatically for you. This is great for when you have pads

and tracks which aren't lined up to your current snap grid. If you don't have these options enabled then you may have to keep reducing your snap grid until you find one that fits. Farmore trouble than it's worth. There is almost never a reason to have these options disabled.

. Always take your track to the center of the pad, don't make your track and pad "just touch". There are few reasons for this. The first is that it's sloppy and unprofessional. The second is that your program may not think that the track is making electrical connection to the pad. Proper use of a snap grid and electrical grid will avoid problems here.

. Use a single track, not multiple tracks tacked together end to end. It may make no difference to the look of your final board, but it can be a pain for future editing. Often you'll have to extend a track a bit. In this case it's best to delete the old one and place a new one. It may take a few extra seconds, but it's worth it. People looking at your finished board may not know, but YOU'LL know! It's the little touches like this that set good PCB designers apart.

. Make sure your tracks go right through the exact center of pads and components, and not off to one side. Use of the correct snap grid will ensure that you get this right every time. If your track doesn't go through the exact center then you are using the wrong snap grid. Why do you need to do this? It makes your board neater and more symmetrical, and it gives you the most clearance.

. Only take one track between 100 thou pads unless absolutely necessary. Only on large and very dense designs should you consider two tracks between pads. Three tracks between pads is not unheard of, but we are talking seriously fine tolerances here.

. For high currents, use multiple vias when going between layers. This will reduce your track impedance and improve the reliability. This is a general rule whenever you need to decrease the impedance of your track or power plane.

. Don't "drag" tracks to angles other than 45 degrees

. "Neck down" between pads where possible. Eg, a 10 thou track through two 60 thou pads gives a generous 15 thou clearance between track and pad.

. If your power and ground tracks are deemed to be critical, then lay them **down** first. Also, make your power tracks as **BIG** as possible.

- . Keep power and ground tracks running in close proximity to each other if possible, don't send them in opposite directions around the board. This lowers the loop inductance of your power system, and allows for effective bypassing.
 - . Keep things symmetrical. Symmetry in tracking and component placement is really nice from a professional aesthetics point of view.
 - . Don't leave any unconnected copper fills (also called "dead copper"), ground them or take them out.
- If you are laying out a non-plated through double sided board, then there are some additional things to watch out for. Non-plate through holes require you to solder a link through the board on both the top and bottom layer.
- . Do not place vias under components. Once the component is soldered in place you won't be able to access the joint to solder a feed through. The solder joint for the feed through can also interfere with the component.
 - . Try and use through hole component legs to connect top tracks to bottom tracks. This minimises the number of vias. Remember that each via adds two solder joints to your board. The more solder joints you have, the less reliable your board becomes. Not to mention that that it takes a lot longer to assemble.

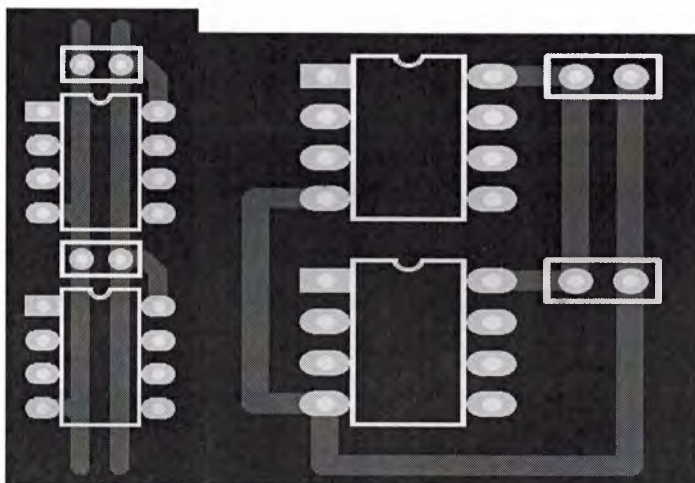


fig:2.3

An example of GOOD power routing (Left) and BAD power routing (Right)

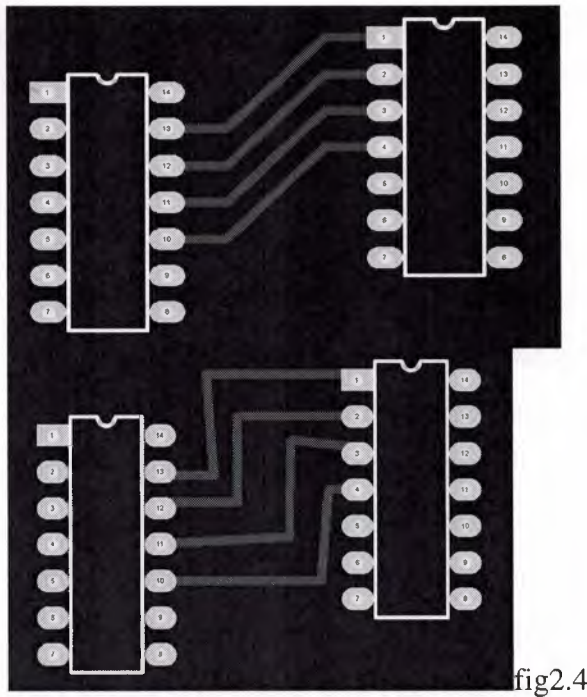


fig2.4

An example of GOOD routing (top) and BAD routing (bottom)

2.2.2 Finishing Touches

Once you have finished all your routing, your board isn't done quite yet. There are a few last minute checks and finishing touches you should do.

If you have thin tracks (<25 thou) then it's nice to add a "chamfer" to any "T" junctions, thus eliminating any 90 degree angles. This makes the track more physically robust, and prevents any potential manufacturing etching problems. But most importantly, it looks nice.



fig:2.5

- . Check that you have any required mounting holes on the board. Keep mounting holes well clear of any components or tracks. Allow room for any washers and screws.
- . Minimise the number of hole sizes. Extra hole sizes cost you money, as the manufacturer will charge you based on not only the number of holes in your boards, but the number of different hole sizes you have. It takes time for the very high speed drill to spin down, change drill bits, and then spin up again. Check with your manufacturer for these costs, but you can't go wrong by minimising the number of hole sizes.
- . Double check for correct hole sizes on all your components. Nothing is more annoying than getting your perfectly laid out board back from the manufacturer, only to find that a component won't fit in the holes! This is a very common problem, don't get caught out.
- . Ensure that all your vias are identical, with the same pad and hole sizes. Remember your pad to hole ratio. Errors here can cause "breakouts" in your via pad, where the hole, if shifted slightly can be outside of your pad. With plated through holes this is not always fatal, but without a complete annular ring around your hole, your via will be mechanically unreliable.
- . Check that there is adequate physical distance between all your components. Watch out for components with exposed metal that can make electrical contact with other components, or exposed tracks and pads.
- . Change your display to "draft" mode, which will display all your tracks and pads as outlines. This will allow you to see your board "warts and all", and will show up any tracks that are tacked on or not ending on pad centers.
- . If you wish, add "teardrops" to all your pads and vias. A teardrop is a nice "smoothing out" of the junction between the track and the pad, not surprisingly, shaped like a teardrop. This gives a more robust and reliable track to pad interface, better than the almost right angle between a standard track and pad. Don't add teardrops manually though, it's a waste of time. But if your program supports automatic teardrop placement, feel free to use it.



fig:2.6

2.2.3 Single Sided Design

Single sided design can greatly reduce the cost of your board. If you can fit your design on a single sided board then it is preferable to do so. Look inside many of today's consumer items like TV's and DVD players, and you will almost certainly find some single sided boards. They are still used because they are so cheap to manufacture. Single sided design however requires some unique techniques which are aren't required once you go to doubled sided and multi-layer design. It is certainly more challenging than a double sided layout. In fact, a single sided board design will be regarded inversely proportional to the number of jumper links used. No jumper links earns the admiration of many peers!

It is all about a balance between board size and the number of jumper links required. Almost every single sided board will require some jumper links, so it is important to minimise these. Component placement is even more critical on a single sided board, so this is no time to make all your components nice and neatly aligned. Arrange your components so that they give the shortest and most efficient tracking possible. It is like playing a game Chess, if you don't think many moves ahead then you will get yourself

in a corner pretty quickly. Having just one track running from one side of your board to the other can ruin your whole layout, as it makes routing any other perpendicular tracks impossible. Many people will route their board as though it is a double sided board, but only with straight tracks on the top layer. Then when the board is to be manufactured, the top layer tracks are replaced with jumper links. This can be a rather inefficient way to approach single sided design, and is not recommended. You must be frugal in your placement, and don't be afraid to rip everything up and try again if you see a better way to

route something. With experience, you will be able to tell before you even start, if a design is worth trying to route on a single sided board.

2.2.4 Double Sided Design

Double sided design gives an extra degree of freedom for designing your board. Things that were next to impossible on a single sided board become relatively easy when you add an additional layer.

Many (inexperienced) designers tend to become lazy when laying out double sided boards. They think that component placement doesn't matter all that much, and hundreds of vias can be used to get them out of trouble. They will often lay out components like ICs in neat rows, and then proceed to route everything using a right angle rules. This means that they will route all the tracks on the bottom layer in one direction, and then all the tracks on the top layer perpendicular to the bottom layer. The theory is that if you chop and change between layers enough times you can route almost anything using a "step" type pattern. This technique can be ugly and inefficient, and is a throw back to the old manual tape days. Many basic auto routers work in this way. Stick to using good component placement techniques and efficient building block routing. Double sided design can also give you the chance to make use of good ground plane techniques, required for high frequency designs. This will be discussed later.

2.2.5 Other Layers

There are a few other important layers on your PCB beside the copper track layers.

2.2.6 Silkscreen

The "silkscreen" layer is also known as the "component overlay" or "component layer". It is the layer on the top of your board (and bottom if needed) that contains your component outlines, designators (C1, R1 etc), and free text.

This is added to your board using a silkscreening process. White is a standard colour, but other colours are available upon request. You can even mix and

match colours on the one board, but that usually costs extra. When designing your board, make sure that you keep all your component designators the same text size, and oriented in the same direction. When laying out your own component footprints, where possible, make sure that you add a component overlay that reflects the actual size of your component. This way you will be able to tell at a glance how close you can physically position your components. Ensure that all polarised components are marked, and that pin 1 is identified.

Your silkscreen layer will be the most inaccurately aligned of all your layers, so don't rely on it for any positional accuracy. Ensure that no part of the silkscreen overlaps a bare pad. There is no minimum width requirement for lines on the component overlay, so feel free to use smaller lines and text sizes to fit things in. If parts of the text or lines don't turn out perfectly on your board then it does not affect your design, unlike tracks and pads. As a general rule, don't put component values on the silkscreen, just the component designator.

2.2.7 Solder Mask

A solder mask is a thin polymer coating on your board which surrounds your pads to help prevent solder from bridging between pins. This is essential for surface mount and fine pitch devices. The solder mask typically covers everything except pads and vias. Your PCB program will automatically remove solder mask from pads and vias. The gap it leaves between the pad and the solder mask is known as the "mask expansion". The mask expansion should usually be set to at least a few thou. Be careful not to make it too big, or there might be no solder mask between very fine pitch devices.

Your solder mask is displayed in your PCB package as a negative image, just like your power plane. Under normal circumstances you don't need to put anything on your solder mask layer. But if you want to leave the solder mask off a certain part of your board, you can place tracks and fills on your solder mask layer. It is often handy to remove a small square of solder mask from the top of your board, where there are no tracks underneath. This leaves a nice bare and visible part of your board to write something with a pen. Solder masks

come in two types, silkscreen, or “photo imageable”. Photo imageable masks provide better resolution and alignment, and are preferred over silkscreened. You can get different colour solder masks, but the standard colour is green. On most standard quality boards, the solder mask is laid directly over the bare copper tracks. This is known as Solder Mask Over Bare Copper, or SMOBC. *You can get other coatings over your tracks in addition to the solder mask, but these are usually for fairly exotic applications.*

You can have vias covered with solder mask if you wish, this is known as tenting. This is useful for close tolerance designs, to prevent solder from flowing into vias.

2.2.8 Mechanical Layer

The mechanical layer (which may go under other names depending on the package) is used to provide an outline for your board, and other manufacturing instructions. It is not part of your actual PCB design, but is very useful to tell the PCB manufacturer how you want your board assembled. There are no hard and fast rules for this layer, use it however you like, just make sure you tell your PCB manufacturer.

2.2.9 Keepout

The keepout layer generally defines areas on your board that you don’t want auto or manually routed. This can include clearance areas around mounting hole pads or high voltage components for instance.

2.2.10 Layer Alignment

When the PCB manufacturer makes your board, there will be alignment tolerances on the artwork film for each layer. This includes track, plane, silkscreen, solder mask, and drilling. If you don’t allow for this in your design, and make your tolerances too fine, you can end up in big trouble. Consult the manufacturer for what alignment tolerances they can achieve, and also what alignment tolerance you are paying for!

2.2.11 Netlists

A netlist is essentially a list of connections (“nets”) which correspond to your schematic. It also contains the list of components, component designators, component footprints and other information related to your schematic. The netlist file can be generated by your schematic package. Generating a netlist is also called “schematic

capture”. Your PCB package can then import this netlist file and do many things. It can automatically load all the required components onto your blank board. It can also assign a “net” name to each of your component pins. With nets assigned to your PCB components, it is now possible to Auto Route, do Design Rule Checking, and display

component connectivity. This is the fundamental concept behind modern Schematic and PCB CAD packages.

2.2.12 Rats Nest

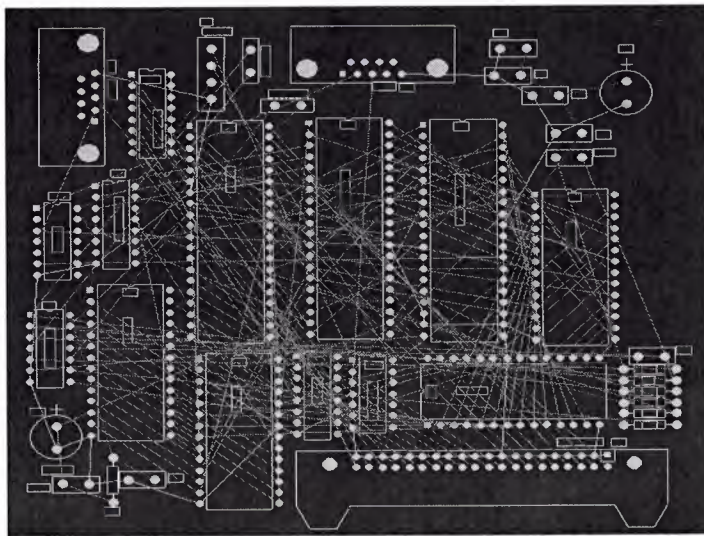


fig:2.7

A Typical “Rats Nest” Display

Your job of component placement will be made infinitely easier by having a “rats nest” display enabled. If there is one reason for going to the trouble of

drawing up an accurate schematic and importing a netlist, this is surely it. For large designs, a rats nest display is essential.

A rats nest display is one where the program will draw a straight line (not a track) between the pads of components which are connected on the schematic. In effect, it shows the connectivity of your circuit before you start laying out tracks. At the start of your board layout, with all your components placed down randomly, this will appear as a huge and complicated random maze of lines. Hence the name rats nest.

The rats nest may look very daunting at first, but when you move each component the lines will automatically move with them. In this way you can see instantly which components are connected to which, without having to refer back to the schematic and constantly cross reference component designators. Once you have used this feature once, you won't want to live without it. Even when doing simple designs with a few dozen components, you will miss this functionality. With the rats nest display enabled, it will be almost possible to lay out all of your components optimally in no time, without having to lay down one single track. The rats nest display will effectively show you what your tracks

will connect to. The rats nest lines should disappear when you route your tracks between components, so your design will get less and less "complicated looking" as you go along. When all the rats nest lines disappear, your board is fully routed.

2.3 DESIGN RULE CHECKING

Design Rule Checking (DRC) allows you to automatically check your PCB design for connectivity, clearance, and other manufacturing errors. With the large and complex PCBs being designed today, it is impractical to manually check a PCB design. This is where the DRC comes into its own, it is an absolutely essential step in professional PCB design.

Examples of what you can check with a DRC are:

- Circuit connectivity. It checks that every track on your board matches the connectivity of your schematic.

Electrical clearance. You can check the clearance between tracks, pads, and components.

Manufacturing tolerances like min/max hole sizes, track widths, via widths, annulus sizes, and short circuits.

A complete DRC is usually performed after you have finished your PCB. Some packages however have the ability to do “real time” (or “online”) DRC checking as you create your board. For instance, it won't let you connect a track to a pad it shouldn't go to, or violate a clearance between track and pad. If you have real-time DRC capability, use it, it's an invaluable tool.

2.3.1 Forward and Back Annotation

Forward Annotation is when you make changes to your existing PCB layout via the schematic editor. The program will take your schematic netlist and component designators, and import them into your PCB design, and making any relevant changes. Some packages will also automatically remove old PCB tracks that are no longer connected. You can do this at any time during your PCB layout. If you update your schematic, then you must forward annotate into your PCB design. You can do edits like this manually, but forward annotation automates the process. Back Annotation is when you change one of the component designators (eg. “C1” to “C2”) on your PCB and then automatically update this information back into your Schematic. More advanced back annotation features allow you to swap gates on chips, and perform other electrical changes. There should never be much real need to use back annotation.

2.3.2 Multi layer Design

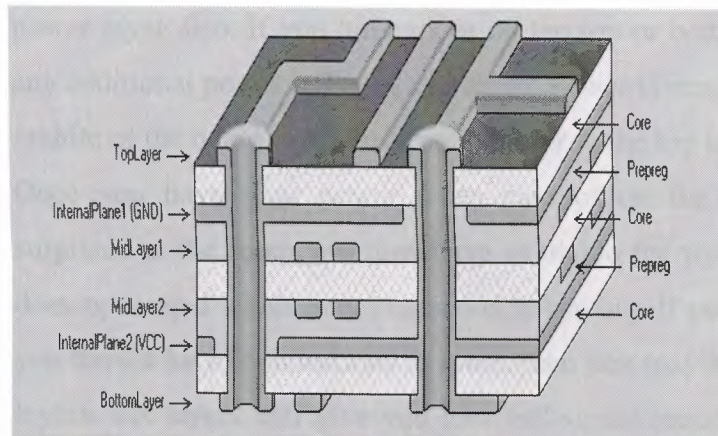


fig:2.8

Typical 6 Layer PCB Construction

A multi layer PCB is much more expensive and difficult to manufacture than a single or double sided board, but it really does give you a lot of extra density to route power and signal tracks. By having your signals running on the inside of your board, you can pack your components more tightly on your board to give you a more compact design.

Deciding to go from double sided to multi layer can be a big decision, so make sure that a multi layer board is warranted on the grounds of board size and complexity. You can forget about making multi layer boards yourself, it requires a commercial manufacturer. Most of the hobby board suppliers will not do multi layer boards. Multi layer boards come in even number of layers. With 4, 6, and 8 layer being the most common. You can go many layers above this, but now you are in the realm of the very specialised.

Technically you can get an odd number of layers manufactured, like a 3 layer board for instance. But it really won't save you any cost over a 4 layer board. In fact a 3 layer board might even be more expensive than a 4 layer board because it calls for a non-standard manufacturing process.

If you decide to go multi layer then make sure you make use all of your layers, there is no point leaving one completely blank. With a multi layer board, you

would typically dedicate one complete layer to a ground plane, and another to your power. With perhaps a few signal tracks thrown on the power layer if you need to. If you have a digital only board, then you'd often dedicate the entire power layer also. If you have room on the top or bottom layer, you can route any additional power rail tracks on there. Power layers are almost always in the middle of the board, with the ground closer to the top layer.

Once you have your power taken care of on the inner layers, you'll be surprised at the room you now have available for your signal tracks. It really does open up a whole new dimension to routing. If power planes are vital, and you have a lot of connections to route, then you may have to move from 4 to 6 layers. Six layers will give you four full signal routing layers and two layers dedicated to power. You can really do some advanced routing with 6 layers. Eight layers and above is basically more of the same. With multi layer design comes the options of using different types of vias to improve your routing density. There are three types of vias - standard, blind, and buried.

Standard vias go through the whole board, and can connect any of the top, bottom or inner layers. These can be wasteful of space on layers which aren't connected.

"Blind" vias go from the outside surface to one of the inner layers only. The hole does not protrude through the other side of the board. The via is in effect "blind" from the other side of the board. "Buried" vias only connect two or more inner layers, with no hole being visible on the outside of the board. So the hole is completely buried inside your board.

Blind and buried vias cost more to manufacture than standard vias. But they are very useful, and almost mandatory for very high density designs like those involving Ball Grid Array (BGA) components.

2.3.3 Power Planes

It is good practice to use "power planes" to distribute power across your board. Using power planes can drastically reduce the power wiring inductance and impedance to your components. This can be vital for high speed digital design for instance. It is good design practice to use power planes whenever possible. They can even be used on double sided boards, if most of your signal tracks are

on the top layer. A power plane is basically one solid copper layer of board dedicated to either your Ground or Power rails, or both. Power planes go in the middle layers of your board, usually on the layers closest to the outer surfaces. On a 4 layer board with complex power requirements it is common to dedicate one layer to your ground plane, and another layer to your various positive and negative power tracks. Your ground rail is usually your signal reference line, so a ground plane is first preference before a power plane is considered.

Many PCB packages have special Power Plane layers that are designed and laid out in reverse to your other normal tracking layers. On a normal tracking layer, your board is assumed to be blank, and you then lay down tracks which will become your actual copper tracks. On a power plane however, your board is assumed to be covered with copper. Laying down tracks on a power plane actually removes the copper. This concept can take some getting used to. A simple power plane will not have any "tracks" (or removed copper bits) at all on it, but will just be one solid layer of copper. In which case you don't need to lay down any tracks to remove any copper. However, it is common practice on more complex boards to "split" the power plane by laying down tracks. This may be done to separate an analog and a digital ground, which will reduce the amount of digital ground noise which is coupled into the more sensitive analog circuitry. A typical split power plane would involve a "track" being placed from near your input power connector or main filter capacitors and the opposite edge of the board. Be careful not to accidentally cause a power "loop" on your board by inadvertently connecting the two halves of your plane on the other side of the board. As a matter of course, you should place "tracks" completely around the outer edge of your board. This will ensure that the power planes do not extend right to the edge of the board. Power planes on the edges of your board can short to not only one another, but also to any guide rails or mounting hardware. You don't have to use the actual Power Plane layer on your PCB package if you don't want to. You can use a regular signal layer and lay down copper fills and tracks yourself. Power Planes layers though often have some advantages that will vary from one PCB package another.

2.3.4 Good Grounding

Grounding is fundamental to the operation of many circuits. Good or bad grounding techniques can make or break your design. There are several grounding techniques which are always good practices to incorporate into any design.

Use copper, and lots of it. The more copper you have in your ground path, the lower the impedance. This is highly desirable for many electrical reasons. Use polygon fills and planes where possible.

Always dedicate one of your planes to ground on multi-layer boards. Make it the layer closest to the top layer.

Run separate ground paths for critical parts of your circuit, back to the main filter capacitor(s). This is known as “star” grounding, because the ground tracks all run out from a central point, often looking like a star. In fact, try and do this as matter of course, even if your components aren’t critical. Separate ground lines keep current and noise from one component from affecting other components.

If using a ground plane, utilise “split” plane techniques to give effective star grounding.

“stitch” required points straight through to your ground plane, don’t use any more track length than you need.

Use multiple vias to decrease your trace impedance to ground.

2.3.5 Good Bypassing

Active components and points in your circuit which draw significant switching current should always be “bypassed”. This is to “smooth” out your power rail going to a particular device. “Bypassing” is using a capacitor across your power rails as physically and electrically close to the desired component or point in your circuit as possible. A typical bypass capacitor value is 100nF, although other values such as 1uF, 10nF and 1nF are often used to bypass different frequencies. You can even have two or three different value capacitors in parallel. When bypassing, you CANNOT replace multiple capacitors with one single capacitor, it defeats the entire purpose of bypassing!

It is not uncommon for a large design to have hundreds of bypass capacitors. As a general rule, you should use at least one bypass capacitor per IC or other switching component if possible. Common values of bypass capacitors are 100nF for general purpose use, 10nF or 1nF for higher frequencies, and 1uF or 10uF for low frequencies. Special low Equivalent Series Resistance (ESR) capacitors are sometimes used on critical designs such as switch mode power supplies.

2.3.6 High Frequency Design Techniques

High frequency design is where you really need to consider the effects of parasitic inductance, capacitance and impedance of your PCB layout. If your signal is too fast, and your track is too long, then the track can take on the properties of a transmission line. If you don't use proper transmission line techniques in these situations then you can start to get reflections and other signal integrity problems. A "critical length" track is one in which the propagation time of the signal starts to get close to the length of the track. On standard FR4 copper boards, a signal will travel roughly 6 inches every nano second. A rule of thumb states that you need to get really concerned when your track length approaches half of this figure. But in reality it can actually be much less than this. Remember that digital square wave signals have a harmonic content, so a 100MHz square wave can actually have signal components extending into the GHz region. In high speed design, the ground plane is fundamental to preserving the integrity of your signals, and also reducing EMI emissions. It allows you to create "controlled impedance" traces, which match your electrical source and load. It also allows you to keep signals coupled "tight" to their return path (ground).

There are many ways to create controlled impedance "transmission" lines on a PCB. But the two most basic and popular ways are called Microstrip and Stripline.

A Microstrip is simply a trace on the top layer, with a ground plane below. The calculation involved to find the characteristic impedance of a Microstrip is relatively complex. It is based on the width and thickness of the

trace, the height above the ground plane, and the relative permittivity of the PCB material. This is why it is important to keep the ground plane as close as possible to (usually) the top layer. A Stripline is similar to the Microstrip, but it has an additional ground plane on top of the trace. So in this case, the trace would have to be on one of the inner layers. The advantage of stripline over microstrip is that most of the EMI radiation will be contained within the ground planes. There are many free programs and spreadsheets available that will calculate all the variations of Microstrip and Stripline for you.

Some useful information and rules of thumb for high frequency design are:

Keep your high frequency signal tracks as short as possible.

Avoid running critical high frequency signal tracks over any cutout in your ground plane. This causes discontinuity in the signal return path, and can lead to EMI problems. Avoid cutouts in your ground plane wherever possible. A cutout is different to a split plane, which is fine, provided you keep your high frequency signal tracks over the relevant continuous plane.

. Have one decoupling capacitor per power pin.

. If possible, track the IC power pin to the bypass capacitor first, and then to the power plane. This will reduce switching noise on your power plane. For very high frequency designs, taking your power pin directly to the power plane provides lower inductance, which may be more beneficial than lower noise on your plane.

. Be aware that vias will cause discontinuities in the characteristic impedance of a transmission line.

To minimise crosstalk between two traces above a ground plane, minimise the distance between the plane and trace, and maximise the distance between traces. The coefficient of coupling between two traces is given by $1/(1+(\text{Distance between traces} / \text{height from plane})^2)$

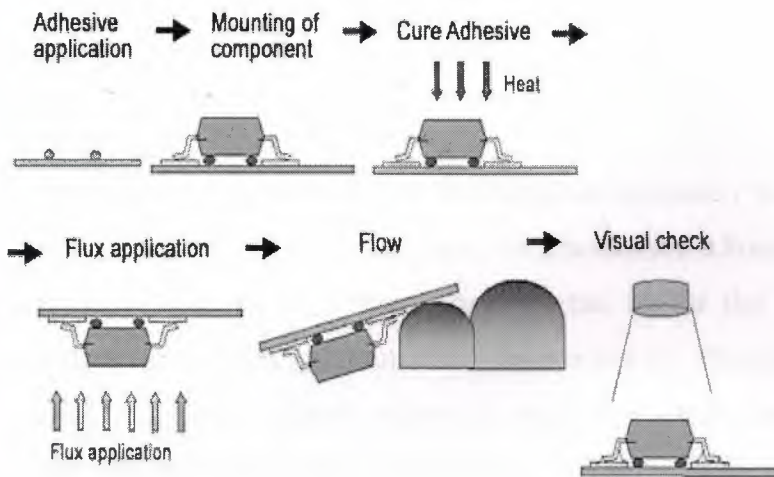
Smaller diameter vias have lower parasitic inductance, and are thus preferred the higher in frequency you go.

. Do not connect your main power input connector directly to your power planes, take it via your main filter capacitor(s).

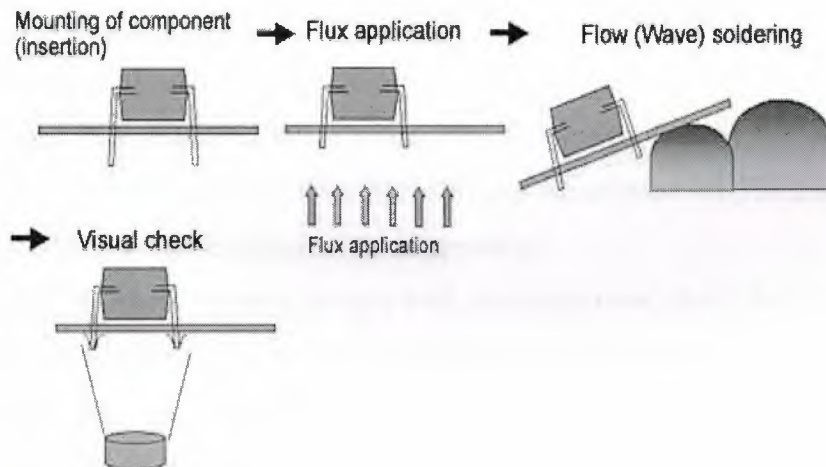
2.3.7 Soldering

Soldering considerations need to be taken into account when laying out your board.

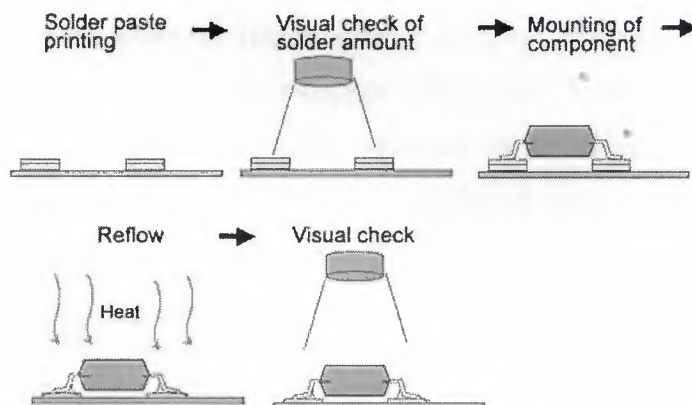
There are three basic soldering techniques - hand, wave, and reflow. Hand soldering is the traditional method typically used for prototypes and small production runs. Major impacts when laying out your board include suitable access for the iron, and thermal relief for pads. Non-plated through double sided boards should allow for ample room to get the soldering iron onto the top side pads. Wave soldering is a common process used for surface mount and through hole production soldering. It involves passing the entire board over a molten bath of solder. Solder masks are absolutely essential here to prevent bridging. The major thing to watch out for when designing is ensuring that small components are not in the wave solder "shadow" of larger components. The board travels through the wave solder machine in one direction, so there will be a lack of solder trailing behind larger components. Surface mount devices are fixed to the board with an adhesive before wave soldering. Reflow soldering is the latest technique, and is suitable for all surface mount components. The blank board is first coated with a mask of solder paste over the pads (solder "stencils" are used for this). Then each component is placed, and is sometimes held in place by an adhesive. The entire board is then loaded into an infrared or nitrogen oven and "baked". The solder paste melts (reflows) on the pads and component leads to make the joint. A newer reflow method called pin-in-paste or intrusive reflow is available for through hole devices. Combinations of wave and reflow soldering can be used for mixed through hole and surface mount boards. Wave soldering has the advantage of being cheap, but the disadvantage of imposing placement limits on your components. Reflow soldering is more complex and expensive, but it allows for very dense surface mount packing.



SMD Wave Soldering



Through-hole Wave Soldering



SMD Reflow Soldering fig:2.9

2.3.8 Basic PCB Manufacture

A PCB usually consists of a blank fibreglass substrate ("the board"), which is usually 1.6mm thick. Other common thicknesses are 0.8mm and 2.4mm. There are many types PCB substrate material, but by far the most common is a standard woven epoxy glass material known as FR4. This material has standard known properties, typical values of which are shown in the accompanying table. The most often used parameter is probably the dielectric constant. This figure is important for calculating high speed transmission line parameters and other effects. An FR4 PCB is made up of glass and resin. Glass has a dielectric constant of approximately 6, and the resin has a dielectric constant of approximately 3. So an FR4 PCB can typically have a figure ranging from under 4, to almost 5. If you need an exact figure you will have to consult with your PCB manufacturer. Typical FR4 Properties: Dielectric Constant 3.9 to 4.8 Dielectric Breakdown 39kV/mm Water Absorption <1.3% Dissipation Factor 0.022 Thermal Expansion 16-19ppm/degC

NOTE: These values can vary with manufacturers, check with your supplier for exact figures. Other exotic base materials like Teflon are also available, but are only used for special designs that require a higher grade base material for a specific reason. There are cheaper materials than FR4, like phenolic base and CEM-1. These are hobbyist grade boards, but are also often used in some mass consumer products due to their low cost. They are not suitable for plated through holes or fine tolerance designs. A blank base material coated with copper is known as a copper clad board. A multi-layer board is made up of various individual boards separated by Preimpregnated Bonding Layers, also known as "prepreg". There are different ways to stack these board layers up, and this will dictate what you can do with planes and blind/buried vias. Consult the manufacturer for their recommendations on this.

2.3.9 Surface Finishies

You can get your PCB manufactured with several different types of pad and track surface finish. Very low cost single and double sided boards without

solder mask typically have a basic tin coated finish. Beware of potential shorts between tracks with this method. Your standard professionally manufactured board will typically have solder mask over bare copper (SMOBC) tracks, and a tin finish on the pads and vias which is Hot Air Leveled (HAL). Hot air leveling helps most surface mount components to sit flat on the board. For large and critical surface mount components, a gold "flash" finish is used on the pads. This gives an extremely flat surface finish for dense fine pitch devices. Peelable solder masks are available, and are handy for temporary masking of areas on your board during wave soldering or conformal coating.

2.3.10 Electrical Testing


You can have your finished PCB checked for electrical continuity and shorts at the time of manufacture. This is done with a automated "flying probe" or "bed of nails" test machine. It checks that the continuity of the tracks matches your PCB file. It may cost a fair bit extra, but this is pretty mandatory for multi-layer boards. *If you have a manufacturing error on one of your inner layers, it can be very difficult to fix.*

2.3.11 Signature

Like any work of art, no board is complete without adding your name or signature to it! The signature can take any form your like. Some people put their name, initials, or a fancy symbol. Whatever it is, just make sure you add something. A signature can be placed on any of the copper layers, or on the component overlay.

3. CIRCUIT ELEMENTS

3.1 RESISTORS

The resistor's function is to reduce the flow of electric current. This symbol  is used to indicate a resistor in a circuit diagram, known as a schematic.

Resistance value is designated in units called the "Ohm." A 1000 Ohm resistor is typically shown as 1K-Ohm (kilo Ohm), and 1000 K-Ohms is written as 1M-Ohm. There are two classes of resistors; fixed resistors and the variable resistors. They are also classified according to the material from which they are made. The typical resistor is made of either carbon film or metal film.

. There are other types as well, but these are the most common. The resistance value of the resistor is not the only thing to consider when selecting a resistor for use in a circuit. The "tolerance" and the electric power ratings of the resistor are also important.

. The tolerance of a resistor denotes how close it is to the actual rated resistance value. For example, a $\pm 5\%$ tolerance would indicate a resistor that is within $\pm 5\%$ of the specified resistance value.

The power rating indicates how much power the resistor can safely tolerate. Just like you wouldn't use a 6 volt flashlight lamp to replace a burned out light in your house, you wouldn't use a 1/8 watt resistor when you should be using a 1/2 watt resistor.

The maximum rated power of the resistor is specified in Watts. Power is calculated using the square of the current (I^2) x the resistance value

R of the resistor. If the maximum rating of the resistor is exceeded, it will become extremely hot, and even burn

Resistors in electronic circuits are typically rated 1/8W, 1/4W, and 1/2W. 1/8W is almost always used in signal circuit applications. When powering a light emitting diode, a comparatively large current flows through the resistor, so you need to consider the power rating of the resistor you choose.

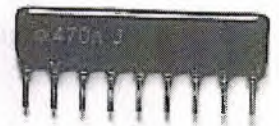
3.2 Types of resistors

3.3 Fixed Resistors

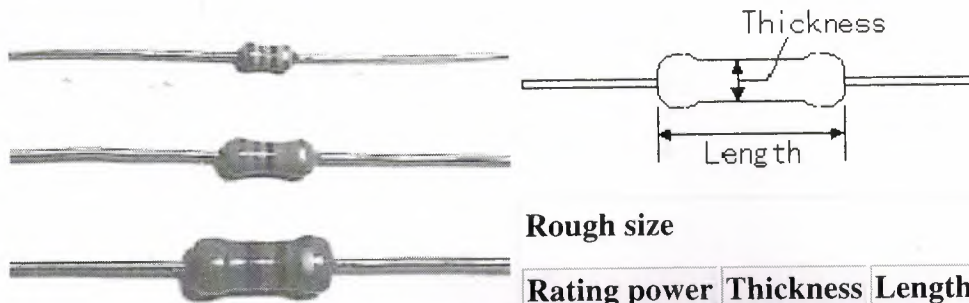
A fixed resistor is one in which the value of its resistance cannot change.

3.4 Carbon film resistors

This is the most general purpose, cheap resistor. Usually the tolerance of the resistance value is $\pm 5\%$. Power ratings of 1/8W, 1/4W and 1/2W are frequently used.



Carbon film resistors have a disadvantage; they tend to be electrically noisy. Metal film resistors are recommended for use in analog circuits. However, we have never experienced any problems with this noise. The physical size of the different resistors are as follows.



From the top of the photograph
 1/8W
 1/4W
 1/2W

Rough size

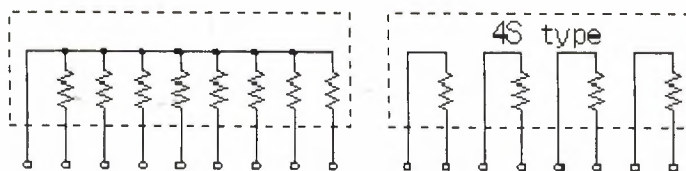
| Rating power (W) | Thickness (mm) | Length (mm) |
|---------------------|-------------------|----------------|
| 1/8 | 2 | 3 |
| 1/4 | 2 | 6 |
| 1/2 | 3 | 9 |

Table:3.1

This resistor is called a Single-In-Line(SIL) resistor network. It is made with many resistors of the same value, all in one package. One side of each resistor is connected with one side of all the other resistors inside. One example of its use would be to control the current in a circuit powering many light emitting diodes (LEDs).

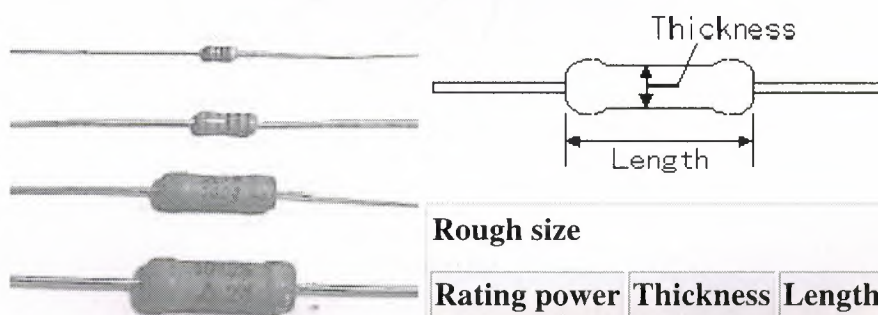
In the photograph on the left, 8 resistors are housed in the package. Each of the leads on the package is one resistor. The ninth lead on the left side is the common lead. The face value of the resistance is printed. (It depends on the supplier.)

Some resistor networks have a "4S" printed on the top of the resistor network. The 4S indicates that the package contains 4 independent resistors that are not wired together inside. The housing has eight leads instead of nine. The internal wiring of these typical resistor networks has been illustrated below. The size (black part) of the resistor network which I have is as follows: For the type with 9 leads, the thickness is 1.8 mm, the height 5mm, and the width 23 mm. For the types with 8 component leads, the thickness is 1.8 mm, the height 5 mm, and the width 20 mm.



3.5 Metal film resistors

Metal film resistors are used when a higher tolerance (more accurate value) is needed. They are much more accurate in value than carbon film resistors. They have about $\pm 0.05\%$ tolerance. They have about $\pm 0.05\%$ tolerance. I don't use any high tolerance resistors in my circuits. Resistors that are about $\pm 1\%$ are more than sufficient. Ni-Cr (Nichrome) seems to be circuits. Resistors that are about $\pm 1\%$ are more than sufficient. Ni-Cr (Nichrome) seems to be used for the material of resistor. The metal film resistor is used for bridge circuits, filter circuits, and low-noise analog signal circuits.



From the top of the photograph

| | |
|------|------------------------|
| 1/8W | (tolerance $\pm 1\%$) |
| 1/4W | (tolerance $\pm 1\%$) |
| 1W | (tolerance $\pm 5\%$) |
| 2W | (tolerance $\pm 5\%$) |

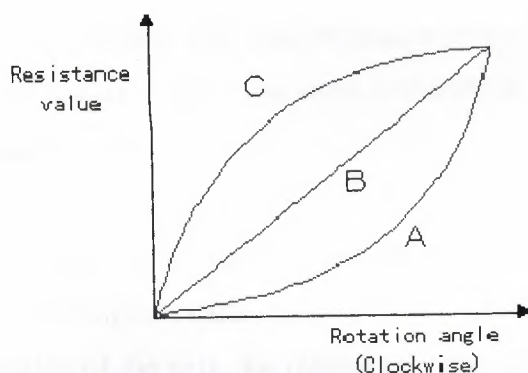
| Rough size | | |
|------------------|----------------|-------------|
| Rating power (W) | Thickness (mm) | Length (mm) |
| 1/8 | 2 | 3 |
| 1/4 | 2 | 6 |
| 1 | 3.5 | 12 |
| 2 | 5 | 15 |

Table:3.2

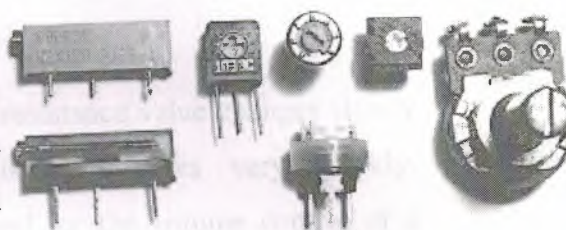
3.6 Variable Resistors


There are two general ways in which variable resistors are used. One is the variable resistor which value is easily changed, like the volume adjustment of Radio. The other is semi-fixed resistor that is not meant to be adjusted by anyone but a technician. It is used to adjust the operating condition of the circuit by the technician. Semi-fixed resistors are used to compensate for the inaccuracies of the resistors, and to fine-tune a circuit. The rotation angle of the variable resistor is usually about 300 degrees. Some variable resistors must be turned many times to use the whole range of resistance they offer. This allows for very precise adjustments of their

value. These are called "Potentiometers" or "Trimmer Potentiometers."



In the photograph to the left, the variable resistor typically used for volume controls can be seen on the far right. Its value is very easy to adjust. The four resistors at the center of the photograph are the semi-fixed type. These ones are mounted on the printed circuit board. The two resistors on the left are the trimmer potentiometers.



This symbol  is used to indicate a variable resistor in a circuit diagram.

There are three ways in which a variable resistor's value can change according to the rotation angle of its axis.

When type "A" rotates clockwise, at first, the resistance value changes slowly and then in the second half of its axis, it changes very quickly. The "A" type variable resistor is typically used for the volume control of a radio, for example. It is well suited to adjust a low sound subtly. It suits the characteristics of the ear. The ear hears low sound changes well, but isn't as sensitive to small changes in loud sounds. A larger change is needed as the volume is increased. These "A" type variable resistors are sometimes called "audio taper" potentiometers.

As for type "B", the rotation of the axis and the change of the resistance value are directly related. The rate of change is the same, or linear, throughout the sweep of the axis. This type suits a resistance value adjustment in a circuit, a balance circuit and so on.

They are sometimes called "linear taper" potentiometers. Type "C" changes exactly the opposite way to type "A". In the early stages of the rotation of the axis, the resistance value changes rapidly, and in the second half, the change occurs more slowly. This type isn't too much used. It is a special use.

As for the variable resistor, most are type "A" or type "B".

3.7 CDS Elements

Some components can change resistance value by changes in the amount of light hitting them. One type is the Cadmium Sulfide Photocell. (Cd) The more light that hits it, the smaller its resistance value



<http://www.birdy-eyes.org>

becomes.

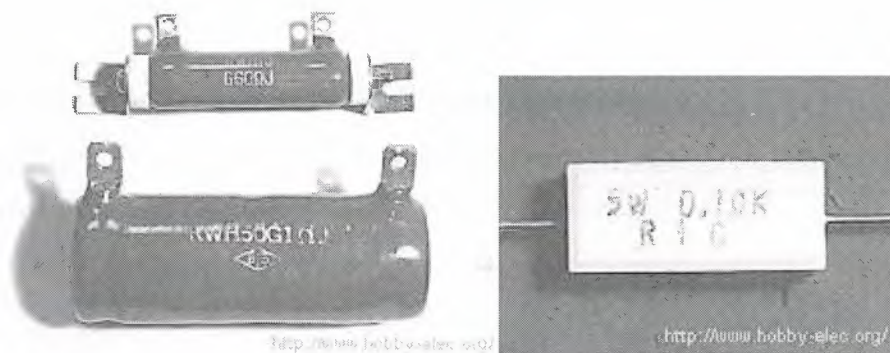
There are many types of these devices. They vary according to light sensitivity, size, resistance value etc

Pictured at the left is a typical CDS photocell. Its diameter is 8 mm, 4 mm high, with a cylinder form. When bright light is hitting it, the value is about 200 ohms, and when in the dark, the resistance value is about 2M ohms. This device is using for the head lamp illumination confirmation device of the car, for example.

3.8 Other Resistors

There is another type of resistor other than the carbon-film type and the metal film resistors.

It is the wirewound resistor. A wirewound resistor is made of metal resistance wire, and because of this, they can be manufactured to precise values. Also, high-wattage resistors can be made by using a thick wire material. Wirewound resistors cannot be used for high-frequency circuits. Coils are used in high frequency circuits. Since a wirewound resistor is a wire wrapped around an insulator, it is also a coil, in a manner of speaking. Using one could change the behavior of the circuit. Still another type of resistor is the Ceramic resistor. These are wirewound resistors in a ceramic case, strengthened with a special cement. They have very high power ratings, from 1 or 2 watts to dozens of watts. These resistors can become extremely hot when used for high power applications, and this must be taken into account when designing the circuit. These devices can easily get hot enough to burn you if you touch one.



The photograph on the left is of wirewound resistors.

The upper one is 10W and is the length of 45 mm, 13 mm thickness

The lower one is 50W and is the length of 75 mm, 29 mm thickness.

The upper one is has metal fittings attached. These devices are insulated with a ceramic coating.



Figure :3.1

The photograph on above is a ceramic (or cement) resistor of 5W and is the height of 9 mm, 9 mm depth, 22 mm width.

3.9 Thermistor (Thermally sensitive resistor)

The resistance value of the thermistor changes according to temperature. This part is used as a temperature sensor. There are mainly three types of thermistor.

3.10 NTC(Negative Temperature Coefficient Thermistor) :

With this type, the resistance value decreases continuously as the temperature rises.

3.11 PTC(Positive Temperature Coefficient Thermistor) :

With this type, the resistance value increases suddenly when the temperature rises above a specific point.

3.12 CTR(Critical Temperature Resister Thermistor) :

With this type, the resistance value decreases suddenly when the temperature rises above a specific point

The NTC type is used for the temperature control.

The relation between the temperature and the resistance value of the NTC type can be calculated using the following formula.

$$R = R_0 \cdot \exp^B \left(\frac{1}{T} - \frac{1}{T_0} \right)$$

R : The resistance value at the temperature T

T : The temperature [K]

R₀ : The resistance value at the reference temperature T₀

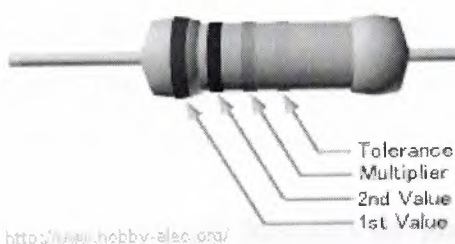
T₀ : The reference temperature [K]

B : The coefficient

Table:3.3

As the reference temperature, typically, 25°C is used. The unit with the temperature is the absolute temperature(Value of which 0 was -273°C) in K(Kelvin).

3.13 Resistor color code



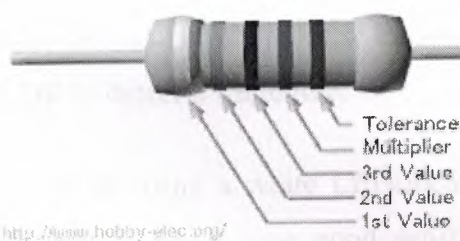
<http://www.hobby-elect.org/>

Example

(Brown=1),(Black=0),(Orange=3)

$$10 \times 10^3 = 10k \text{ ohm}$$

$$\text{Tolerance(Gold)} = \pm 5\%$$



<http://www.hobby-elect.org/>

Example

(Yellow=4),(Violet=7),(Black=0),(Red=2)

$$470 \times 10^2 = 47k \text{ ohm}$$

$$\text{Tolerance(Brown)} = \pm 1\%$$

| Color | Value | Multiplier | Tolerance (%) |
|--------|-------|------------|---------------|
| Black | 0 | 0 | - |
| Brown | 1 | 1 | ±1 |
| Red | 2 | 2 | ±2 |
| Orange | 3 | 3 | ±0.05 |
| Yellow | 4 | 4 | - |
| Green | 5 | 5 | ±0.5 |
| Blue | 6 | 6 | ±0.25 |
| Violet | 7 | 7 | ±0.1 |
| Gray | 8 | 8 | - |
| White | 9 | 9 | - |
| Gold | - | -1 | ±5 |
| Silver | - | -2 | ±10 |
| None | - | - | ±20 |

table:3.4

3.14 Sensor - LDR

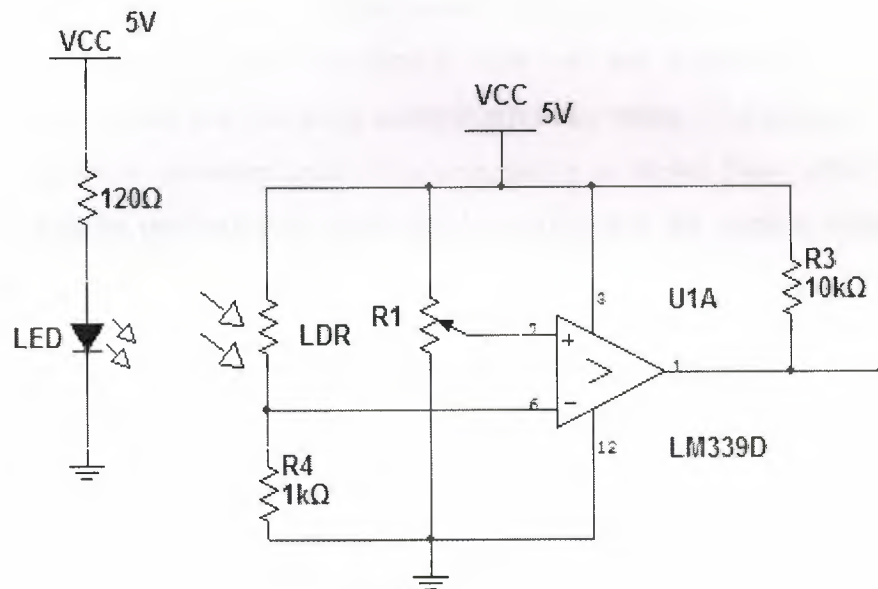
Sensors help the robots become autonomous. Sensors, are just like eyes, ears and skin of the robot. They give your robot the necessary tools to sense its environment and decide on its own what to do next. Depending on your applications, there are different sensors available. Some of the common ones used by hobbyists are LDRs, IR detecting diodes, modulated IR receivers, Ultrasound sensors and the LASER sensors.

LDRs and the IR detecting diodes are the ones that have short range and are usually used in robots like a line follower which does not require a longer range. They are relatively cheap and easy to work with. So, as the topic name goes, I will be describing about using LDRs today.

Light Dependant Resistors or photoresistors or photocells are devices that change their resistance when light falls on them. When there is no light, it will have very high resistance. As the light intensity increases, its resistance decreases. Hence, we can use a LED and focus the light on to a surface that you want to detect. Place a LDR in the path of reflection. Thus, depending on the intensity of the reflected light, we will have different resistance and hence, the robot will be able to sense its surroundings. Lets see how to use the LED and LDR to detect a white line.

I will be using a white LED(meaning it will emit white light) in this post. White LED will give good sensing ability for the fact that white light is reflected fully by a white line. So, we will be able to sense the white line distinctly using a white LED. Now, the one thing you must be aware before going into building the circuit is that the output of the sensor you get will be analog in nature. So, if you are using this output to drive any IC, you will need to convert this to a digital signal first. That can be done in two ways, using comparators or using ADCs(Analog-to-Digital Converters). I will be using the comparator as its very easy to handle and also cheap. After the comparator, you

will get a signal that can be fed to any IC, be it a micro-controller or logic gates.



LDR sensor circuit

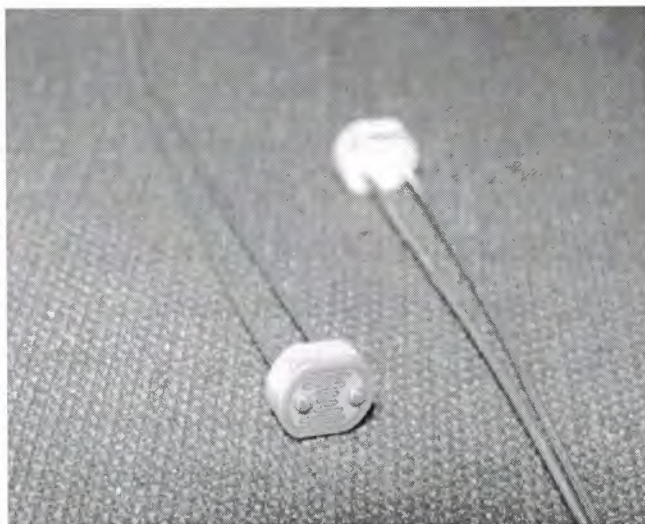


fig:3.2

LDR sensors

The above circuit shows a simple design of sensor circuit. Towards the left is the emitter part. The 120Ω resistance is to limit current through the diode. You

can make the LED grow brighter or dimmer by changing the value of resistance.

Now, let's see what happens in the LDR circuit part. The comparator used here is the ever popular LM339D. It's a quad comparator IC package. It has four comparators and here only one of them is shown. The potentiometer R1 acts as the reference voltage for the comparator. You can use a potentiometer of 10k Ω /10 turns so that you can set a suitable reference value. The output from the LDR is given to the other input of the comparator as shown. Now, after you set up the circuit (preferably on your bread board) check the various voltage levels once.

See the voltage at the center pin of the pot (potentiometer) that is given to the pin 7 of LM339D. Now, by turning the pot you will be able to vary this voltage. Set it to 2.5V initially. Now, you check the voltage at pin 6 that is due to the LDR. First, when the LDR is covered, you will see that the voltage is less. Depending on the lighting conditions, it must be well within 1V. Now, allow the light to fall on the LDR and see the rise in the voltage. It will be around 4 - 4.5 V. Now, that you have tested your inputs, it's time to see if you are getting the proper output.

The resistance R3 at the output is called the pull up resistor. The chip provides an output of only ground. Hence, you pull up the output line to high using this resistor unless the IC pulls it down. Now, when the light falls on the LDR, the input at pin 6 will be less than the reference voltage which we have set at 2.5V. Now, the output will be low. If you take the light off the LDR, the voltage at pin 6 rises above the reference. Hence, the output will be high. Thus, we get a digital output from the circuit. Now, one more good thing about this circuit is that you can set your reference voltage level by varying the pot. This will help to operate the sensor in any environment. All you need to do is check the minimum and maximum voltage from the LDR as described before and set the reference voltage value somewhere in between the two values obtained. That's how you calibrate your sensors to a given environment.

3.2 OPERATIONAL AMPLIFIER

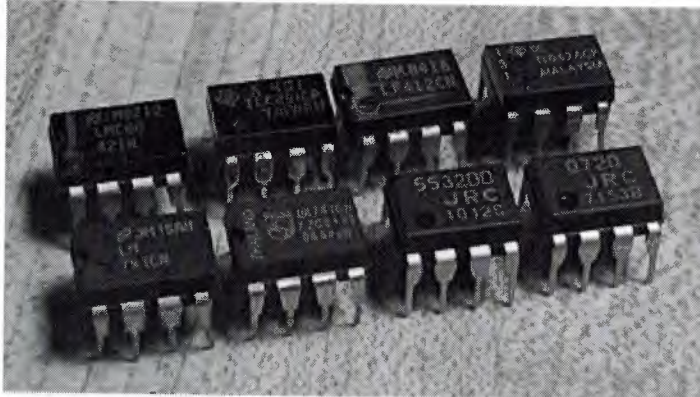


fig:3.3

An operational amplifier, often called an op-amp, is a DC-coupled high-gain electronic voltage amplifier with differential inputs^[1] and, usually, a single output. Typically the output of the op-amp is controlled either by negative feedback, which largely determines the magnitude of its output voltage gain, or by positive feedback, which facilitates regenerative gain and oscillation. High input impedance at the input terminals and low output impedance are important typical characteristics.

Op-amps are among the most widely used electronic devices today, being used in a vast array of consumer, industrial, and scientific devices. Many standard IC op-amps cost only a few cents in moderate production volume; however some integrated or hybrid operational amplifiers with special performance specifications may cost over \$100 US in small quantities.

Modern designs are electronically more rugged than earlier implementations and some can sustain direct short-circuits on their outputs without damage.

3.2.1 History

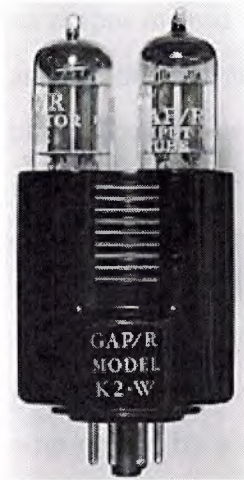


fig:3.4

GAP/R's K2-W: a vacuum-tube op-amp (1953)

GAP/R's model P45: a solid-state, discrete op-amp (1961)

GAP/R's model PP65: a solid-state op-amp in a potted module (1962)

ADI's HOS-050: a high speed hybrid IC op-amp (1979)



fig:3.5

An op-amp in a modern DIP package

An op-amp, defined as a general-purpose, DC-coupled, high gain, inverting feedback amplifier, is first found in US Patent 2,401,779 "Summing Amplifier" filed by Karl D. Swartzel Jr. of Bell labs in 1941. This design used three vacuum tubes to achieve a gain of 90dB and operated on voltage rails of $\pm 350\text{V}$. In contrast to modern day op-amps, it had a single inverting input and an output instead of the modern two differential inputs where one is inverting and the other is not. Throughout WWII, Swartzel's design proved its value by being liberally used in the M9 artillery director designed at Bell Labs. This artillery director worked with the SCR584 radar system to achieve extraordinary hit rates (near 90%) that would not have been possible otherwise.

In 1947, the op-amp was first formally defined and named in a paper by Professor John R. Ragazzoni. In this same paper a footnote mentioned an op-amp design by a student that would turn out to be quite significant. This op-amp, designed by Loebe Julie, was superior in a variety of ways. It had two major innovations. Its input stage used a long-tailed triode pair with loads matched to reduce drift in the output and, far more importantly, it was the first op-amp design to have two inputs (both inverting and non-inverting). The differential input made a whole range of new functionality possible, but it would not be used for a long time due to the rise of the chopper-stabilized amplifier.

In 1949, Edwin A. Goldberg, designed a chopper stabilized op-amp. This set-up uses a normal op-amp with an additional AC amplifier that goes alongside the op-amp. The chopper gets an AC signal from DC by switching between the DC voltage and ground at a fast rate (60Hz or 400Hz). This signal is then amplified, rectified, filtered and fed into the op-amp's non-inverting input. This vastly improved the gain of the op-amp while significantly reducing the output drift and DC offset. Unfortunately, any design that used a chopper couldn't use their non-inverting input for any other purpose. Nevertheless, the much improved characteristics of the chopper-stabilized op-amp made it the dominant way to use op-amps. Techniques that used the non-inverting input regularly would not be very popular until the 1960s when op-amp ICs started to show up in the field.

In 1953, vacuum tube op-amps became commercially available with the release of the K2-W from GAP/R. It sold in an octal package and had a (K2-P) chopper add-on available that would effectively "use up" the non-inverting input. This op-amp was based on a descendant of Loebe Julie's 1947 design and, along with its successors, would start the widespread use of op-amps in industry.

With the birth of the transistor in 1947, and the silicon transistor in 1954, the concept of ICs became a reality. The introduction of the planar process in 1959 made transistors and ICs stable enough to be commercially useful. By 1961, solid-state, discrete op-amps were being produced. These op-amps were effectively small circuit boards with packages such as edge-connectors. They usually had hand-selected resistors in order to improve things such as voltage offset and drift. The P45 (1961) had a gain of 94dB and ran on $\pm 15\text{V}$ rails. It was intended to deal with signals in the range of $\pm 10\text{V}$. By 1962, several companies were producing modular potted packages that could be plugged into printed circuit boards. These packages were crucially important as they made the operational amplifier into a single black box which could be easily treated as a component in a larger circuit.

In 1963, the first monolithic IC op-amp, the $\mu\text{A}702$ designed by Bob Widlar at Fairchild Semiconductor, was released. Monolithic ICs consist of a single chip as opposed to a chip and discrete parts (a discrete IC) or multiple chips bonded and connected on a circuit board (a hybrid IC). Almost all modern op-amps are monolithic ICs; however, this first IC did not meet with much success. Issues such as an uneven supply voltage, low gain and a small dynamic range held off the dominance of monolithic op-amps until 1965 when the $\mu\text{A}709^{[2]}$ (also designed by Bob Widlar) was released. The popularity of monolithic op-amps was further improved upon the release of the LM101 in 1967, which solved a variety of issues and the subsequent release of the $\mu\text{A}741$ in 1968. The $\mu\text{A}741$ was extremely similar to the LM101 except that Fairchild's facilities allowed them to include a 30pF compensation capacitor inside the chip instead of requiring external compensation. This simple difference has made the 741 the canonical op-amp and many modern amps base their pinout on the 741's. The

μ A741 is still in production, and has become ubiquitous in electronics — many manufacturers produce a version of this classic chip, recognizable by part numbers containing "741."

Since the 741, there have been many different directions taken in op-amp design. Varactor bridge op-amps started to be produced in the late 1960s; they were designed to have extremely small input current and are still amongst the best op-amps available in terms of common-mode rejection with the ability to correctly deal with 100s of Volts at its inputs. In the 1970s high speed, low-input current designs started to be made by using FETs. These would be largely replaced by op-amps made with MOSFETs in the 1980s. During the 1970s single sided supply op-amps also became available. The LM324 (released in 1972) was one such op-amp that came in a quad package and became an industry standard. In addition to packaging multiple op-amps in a single package, the 1970s also saw the birth of op-amps in hybrid packages. These op-amps were generally improved versions of existing monolithic op-amps and were without a doubt the best op-amps available. As the properties of monolithic op-amps improved, the more complex hybrid ICs were quickly relegated to systems that are required to have extremely long service lives or other specialty systems.

Timeline

- 1941: First (vacuum tube) op-amp
- 1947: First op-amp with an explicit non-inverting input
- 1948: First chopper-stabilized op-amp⁹
- 1961: First discrete IC op-amps
- 1962: First op-amps in potted modules
- 1963: First monolithic IC op-amp
- 1966: First varactor bridge op-amps
- 1968: Release of the μ A741 - would be seen as a nearly ubiquitous chip
- 1970: First high-speed, low-input current FET design
- 1972: Single sided supply op-amps being produced

3.2.2 Operation of ideal op-amps

The amplifier's differential inputs consist of an inverting input and a non-inverting input, and ideally the op-amp amplifies only the difference in voltage between the two. This is called the "differential input voltage". In its most common use, the op-amp's output voltage is controlled by feeding a fraction of the output signal back to the inverting input. This is known as negative feedback. If that fraction is zero, i.e., there is no negative feedback, the amplifier is said to be running "open loop" and its output is the differential input voltage multiplied by the total gain of the amplifier, as shown by the following equation:

where V_+ is the voltage at the non-inverting terminal, V_- is the voltage at the inverting terminal and G is the total open-loop gain of the amplifier.

Because the magnitude of the open-loop gain is typically very large and not well controlled by the manufacturing process, op-amps are not usually used without negative feedback. Unless the differential input voltage is extremely small, open-loop operation results in op-amp saturation (see below in Nonlinear imperfections). An example of how the output voltage is calculated when negative feedback exists is shown below in Basic non-inverting amplifier circuit.

Another typical configuration of op-amps is the positive feedback, which takes a fraction of the output signal back to the non-inverting input. An important application of it is the comparator with hysteresis (see Schmitt trigger).

For any input voltages the ideal op-amp has

- infinite open-loop gain,

- infinite bandwidth,
- infinite input impedances (resulting in zero input currents),
- zero offset voltage,
- infinite slew rate,
- zero output impedance, and
- zero noise.

The inputs of an ideal op-amp under negative feedback can be modeled using a nullator, the output with a norator and the combination (complete ideal op-amp) by a nullor.

3.2.3 Applications

3.2.4 Use in electronics system design

The use of op-amps as circuit blocks is much easier and clearer than specifying all their individual circuit elements (transistors, resistors, etc.), whether the amplifiers used are integrated or discrete. In the first approximation op-amps can be used as if they were ideal differential gain blocks; at a later stage limits can be placed on the acceptable range of parameters for each op-amp.

Circuit design follows the same lines for all electronic circuits. A specification is drawn up governing what the circuit is required to do, with allowable limits. For example, the gain may be required to be 100 times, with a tolerance of 5% but drift of less than 1% in a specified temperature range; the input impedance not less than 1 megohm; etc.

A basic circuit is designed, often with the help of circuit modeling (on a computer). Specific commercially available op-amps and other components are then chosen that meet the design criteria within the specified tolerances at acceptable cost. If not all criteria can be met, the specification may need to be modified.

A prototype is then built and tested; changes to meet or improve the specification, alter functionality, or reduce the cost, may be made.

Basic non-inverting amplifier circuit

The general op-amp has two inputs and one output. The output voltage is a multiple of the difference between the two inputs (some are made with floating, differential outputs):

$$V_{\text{out}} = G(V_+ - V_-)$$

G is the open-loop gain of the op-amp. The inputs are assumed to have very high impedance; negligible current will flow into or out of the inputs. Op-amp outputs have very low source impedance.

If the output is connected to the inverting input, after being scaled by a voltage divider $K = R_1 / (R_1 + R_2)$, then:

$$V_+ = V_{\text{in}}$$

$$V_- = K V_{\text{out}}$$

$$V_{\text{out}} = G(V_{\text{in}} - K V_{\text{out}})$$

Solving for $V_{\text{out}} / V_{\text{in}}$, we see that the result is a linear amplifier with gain:

$$V_{\text{out}}/V_{\text{in}} = G / (1 + G K)$$

If G is very large, $V_{\text{out}}/V_{\text{in}}$ comes close to $1/K$, which equals $1 + (R_2/R_1)$.

This negative feedback connection is the most typical use of an op-amp, but many different configurations are possible, making it one of the most versatile of all electronic building blocks.

When connected in a negative feedback configuration, the op-amp will try to make V_{out} whatever voltage is necessary to make the input voltages as nearly

equal as possible. This, and the high input impedance, are sometimes called the two "golden rules" of op-amp design (for circuits that use negative feedback):


1. No current will flow into the inputs.
2. The input voltages will be nearly equal.


The exception is if the voltage required is greater than the op-amp's supply, in which case the output signal stops near the power supply rails, V_{S+} or V_{S-} .

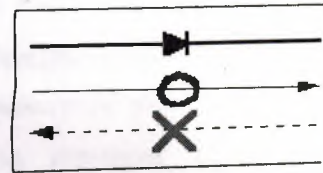
Most single, dual and quad op-amps available have a standardized pin-out which permits one type to be substituted for another without wiring changes. A specific op-amp may be chosen for its open loop gain, bandwidth, noise performance, input impedance, power consumption, or a compromise between any of these factors.

Other applications

- audio- and video-frequency pre-amplifiers and buffers
- voltage comparators
- differential amplifiers
- differentiators and integrators
- filters
- precision rectifiers
- precision peak detectors
- voltage and current regulators
- analog calculators
- analog-to-digital converters
- digital-to-analog converter
- voltage clamps
- oscillators and waveform generators

Some ways in which the diode can be used are listed here. A diode can be used as a rectifier that converts AC (Alternating Current) to DC (Direct Current) for a power supply device. Diodes can be used to separate the signal from radio frequencies. Diodes can be used as an on/off switch that controls current. This symbol  is used to indicate a diode in a circuit diagram.

The meaning of the symbol is (Anode)  (Cathode). Current flows from the anode side to the cathode side.




Although all diodes operate with the same general principle, there are different types suited to different applications. For example, the following devices are best used for the applications noted.

3.3.1 Types Of Diodes Voltage regulation diode (Zener Diode)

The circuit symbol is .

It is used to regulate voltage, by taking advantage of the fact that Zener diodes tend to stabilize at a certain voltage when that voltage is applied in the opposite direction.

3.3.2 Light emitting diode

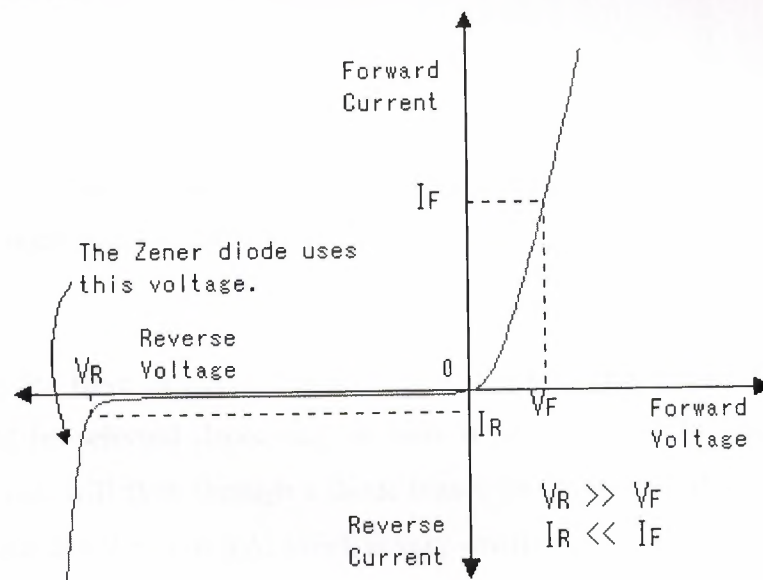
The circuit symbol is .

This type of diode emits light when current flows through it in the forward direction. (Forward biased.)

3.3.3 Variable capacitance diode

The circuit symbol is .

The current does not flow when applying the voltage of the opposite direction to the diode. In this condition, the diode has a capacitance like the capacitor. It is a very small capacitance. The capacitance of the diode changes when changing voltage. With the change of this capacitance, the frequency of the oscillator can be changed. The graph on the right shows the electrical



characteristics of a typical diode.

When a small voltage is applied to the diode in the forward direction, current flows easily.

Because the diode has a certain amount of resistance, the voltage will drop slightly as current flows through the diode. A typical diode causes a voltage drop of about 0.6 - 1V (V_F) (In the case of silicon diode, almost 0.6V) This voltage drop needs to be taken into consideration in a circuit which uses many diodes in series. Also, the amount of current passing through the diodes must be considered.

When voltage is applied in the reverse direction through a diode, the diode will have a great resistance to current flow.

Different diodes have different characteristics when reverse-biased. A given diode should be selected depending on how it will be used in the circuit. The current that will flow through a diode biased in the reverse direction will vary from several mA to just μA , which is very small.

The limiting voltages and currents permissible must be considered on a case by case basis. For example, when using diodes for rectification, part of the time they will be required to withstand a reverse voltage. If the diodes are not chosen carefully, they will break down.

3.3.4 Rectification / Switching /

Regulation diode

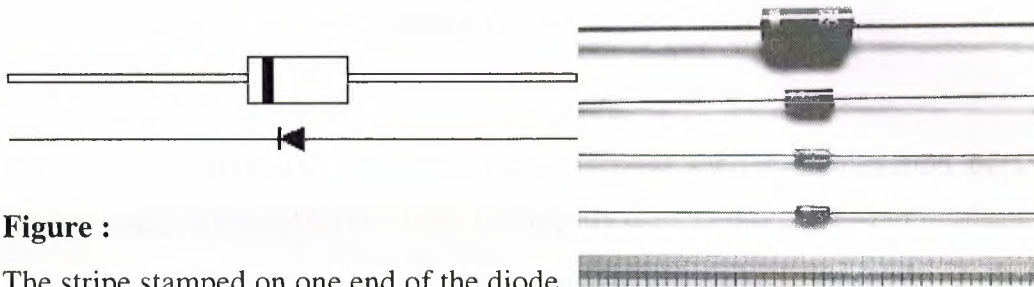


Figure :

The stripe stamped on one end of the diode shows indicates the polarity of the diode.

Fig:3.6

The stripe shows the cathode side.

The top two devices shown in the picture are diodes used for rectification. They are made to handle relatively high currents. The device on top can handle as high as 6A, and the one below it can safely handle up to 1A. However, it is best used at about 70% of its rating because this current value is a maximum rating.

The third device from the top (red color) has a part number of 1S1588. This diode is used for switching, because it can switch on and off at very high speed. However, the maximum current it can handle is 120 mA. This makes it well suited to use within digital circuits. The maximum reverse voltage (reverse bias) this diode can handle is 30V.

The device at the bottom of the picture is a voltage regulation diode with a rating of 6V. When this type of diode is reverse biased, it will resist changes in

voltage. If the input voltage is increased, the output voltage will not change. (Or any change will be an insignificant amount.) While the output voltage does not increase with an increase in input voltage, the output current will. This requires some thought for a protection circuit so that too much current does not flow.

The rated current limit for the device is 30 mA.

Generally, a 3-terminal voltage regulator is used for the stabilization of a power supply. Therefore, this diode is typically used to protect the circuit from momentary voltage spikes. 3 terminal regulators use voltage regulation diodes inside.

3.3.5 Light Emitting Diode (LED)

Light emitting diodes must be choosen according to how they will be used, because there are various kinds.

The diodes are available in several colors. The most common colors are red and green, but there are even blue ones.

The device on the far right in the photograph combines a red LED and green LED in one package. The component lead in the middle is common to both LEDs. As for the remaing two leads, one side is for the green, the other for the red LED. When both are turned on simultaneously, it becomes orange.

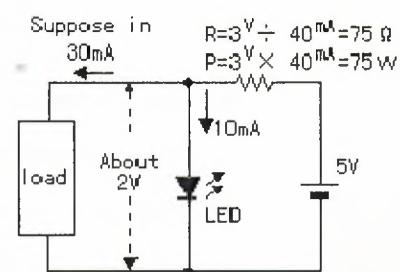


Figure:3.7

When an LED is new out of the package, the polarity of the device can be

determined by looking at the leads. The longer lead is the Anode side, and the short one is the Cathode side.

The polarity of an LED can also be determined using a resistance meter, or even a 1.5 V battery.

When using a test meter to determine polarity, set the meter to a low resistance measurement range. Connect the probes of the meter to the LED. If the polarity is correct, the LED will glow. If the LED does not glow, switch the meter probes to the opposite leads on the LED. In either case, the side of the diode which is connected to the black meter probe when the LED glows, is the Anode side. Positive voltage flows out of the black probe when the meter is set to measure resistance.

It is possible to use an LED to obtain a fixed voltage. The voltage drop (forward voltage, or V_F) of an LED is comparatively stable at just about 2V.



<http://www.hobby-led.org>

3.3.6 Shottky barrier diode

Diodes are used to rectify alternating current into direct current. However, rectification will not occur when the frequency of the alternating current is too high. This is due to what is known as the "reverse recovery characteristic." The reverse recovery characteristic can be explained as follows: IF the opposite voltage is suddenly applied to a forward-biased diode, current will continue to flow in the forward direction for a brief moment. This time until the current stops flowing is called the Reverse Recovery Time. The

current is considered to be stopped when it falls to about 10% of the value of the peak reverse current.

The Schottky barrier diode has a short reverse recovery time, which makes it ideally suited to use in high frequency rectification.

The Schottky barrier diode has the following characteristics. The voltage drop in the forward direction is low. The reverse recovery time is short. However, it has the following disadvantages. The diode can have relatively high leakage current. The surge resistance is low.

CONCLUSION

Fully custom designed sensor circuits have been used in this PLC controlled greenhouse automation project. The first control unit is the section that automatically switches on the ventilation system based on the greenhouses internal temperature. The second control unit is the section that switches on the ultra-violet illumination system based on the external light sensitivity. And the third section is the unit that automatically irrigates and adjust the humidity level based on the humidity level of the soil.

The sensors work with 12v power, while the output voltage is adjusted for the PLC input that requires 24v through optocoupler. Control is maintained according to the signals received from the sensors via the plc. Op-amp (lm-741) has been used as a comparator at the sensors.

PTC has been used the temperature control-unit. The following is the working principle of the thermal control unit;

When the internal environment temperature increases and the op-amps (+) end becomes active and the output becomes (1) consequently, the cooling system becomes active via plc control.

LDR has been used for the illumination section. When the amount of light at the sensor falls below the desired level, the LDR's resistance increases and the op-amp's (+) end becomes active and the output becomes (1) consequently the ultraviolet illumination system is turned on via plc control.

For irrigation controlled by probes which are located on the greenhouse ground serve as sensors. The sensor measures the humidity level of the soil by benefiting from the water's conductivity property. The resistance of the sensor increases as the humidity level decreases and the op-amp's positive end becomes active. The irrigation system is operated by the plc.

References

- [1].<http://www.knx.org/fileadmin/downloads/05%20--%20KNX%20Partners20Becoming%20a%20KNX%20Scientific%20Partner/>
- [2].http://%20Conference%20Papers%20Pisa/18_Greenhouse%20climate%20control.pdf
- [2]. http://www.actahort.org/books/718/718_19.htm
- [3]. <http://www.plcmanual.com/troubleshooting-ii>
- [4].http://www.eod.gvsu.edu/~jackh/books/plcs/chapters/plc_intro.pdf
- [5].http://en.wikipedia.org/wiki/Operational_amplifier
- [6].<http://cognoscis.wordpress.com/2008/05/01/sensor-ldr/>
- [7]<http://www.hobyelectronics.com>