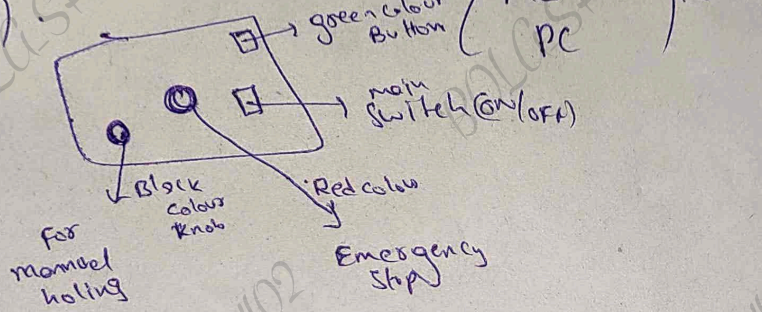
1) Acrylic bit

2) Hand held remote.

3) Auto clock cable

4)



2)

tor is there

or.

25000 RPM.

50 mm/sec

or 3500 mm/min

0.0035 mm

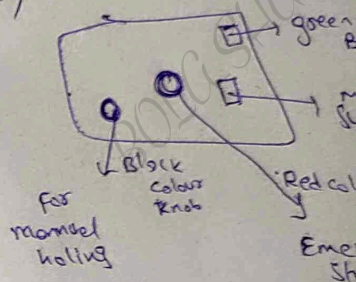
Clean the copper plate

Machine Model

Machine specification

1) PCB Mate 300 watt.
2) This is 3 axis machine (X, Y, Z)
3) Maximum working area
   X-axis — 200mm
   Y-axis — 300mm
4) It will be working for two method
   (i) Manual Mode
   (ii) PC Mode. (system mode)
5) Every axis individual stepper motor is there
6) Spindle motor → BLDC motor.
   ↓ Rotation speed is 25000 RPM.
   X & Y axis. working speed is 58 mm/sec @ 3500 mm/min
   ↓ Machine resolution → 0.0035 mm

1) Acrylic bit
2) Hand held rem
3) Auto check. cable
4)

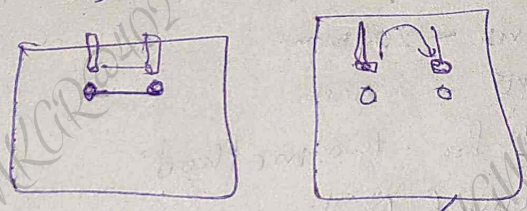


Machine process

1. Engraving
2. Masing
3. drill
4. cut

Z = +2 reason.

one drill completed and then 2mm drill goes up and goes to distance



X

Machine access area

X - 200 mm
Y - 300 mm

- Copper cad
  ↓
  industorial tape
  ↓
  remove double sided tape.

30° at 1mm engraving bit Red colour box.

Drill → Drill box (white colour)

CUT → cut box (blue box)

- Left Side: 13mm spanner
- Right Side: 17mm spanner

Auto leveler

Software – 3

- checking for copper cad is linear / non linear

procedure ↓

- linux CAN —change→ Mach-3mill
- Browse for G-code
- goto in ur circuit
- select engraving.txt file
- click & open
- directly give to create level g-code
- AL.engraving.tab file (Text file → tab file).

(location)

Mach-3-mill

Software-4

open → Reset-enable

ii) off the reset button and

iii) give X=0, Y=0, Z=2

- File → load G-code

take from ur folder

AL engraving file take

Machine process

1. Engraving
2. Masking
3. drill
4. cut

Z = +2 reason

one drill comp

2mm drill g

distance

X

- copper cad

industrial tape

remove dau

- Left side

13mm spanner

Right side

17mm spa

- machine → mill
  ↓
  Drill (1.0) → click ok
  ↓
  G00 Z2
  G00 X00 Y00
  ↓
  file → save as
  ↓
  saved in some location (Bridge rectifier)
  ↓
  filename → 1.0 DRILL

- machine → mill
  ↓
  Drill (1.1) → click ok
  ↓
  G00 Z2
  G00 X00 Y00
  ↓
  file save as
  ↓
  saved in Bridge Rectifier
  ↓
  filename → 1.1 DRILL

Now engraving select ok

↓

Gcode generated

In program
After G00 Z2 line
G00 Z2
G00 X00 Y00

Now circuit select ok

↓

Gcode.

↓

Save
in the same file
you have taken

Filename ENGRAVING

- goto machine → mill

↓

cut layer (ok)

↓

G00 Z2

G00 X00 Y00.

↓

File Save as

↓

Saved in some location

Filename CUT

- machine → mill

↓

Drill (0.8) (click ok)

↓

G00 Z2

G00 X00 Y00.

↓

File → save as

↓

Saved in some location (bridge rectifier)

↓

Filename 0.8 DRILL

- machine → mill

↓

Drill (

↓

G00 Z

G00 X

↓

File →

↓

Saved i

↓

Filena

- Machine → mill

↓

Drill (

↓

G00

G00

↓

File

↓

Save

↓

Filenam

step ④ Selected tool

→ goto parameters → selected tool

List of Drill

(one Page opened)
↓
only change
List of DRILLS.
- close befor drills
- select 0.8 drill
- select 1.0 drill
- select 1.1 drill

↓
click ok.

step ⑤ List of drill

→ goto Machine → list of drill
⇒ just for matched or not matched checking purpose.

step ⑥ Contour

→ goto Machine → · contours
↓
Calculate contour
↓
Give ok
(it gives final prototype board view)
(given board analizing view)

step ⑦ G-Code generation.

→ goto Machine → select Mill → · shows one pop up
output file
↓
select ☑ Section #1

output
☑ Section #1

Any circuit · engraving & circuit & 3 drill (0.8, 1.0, 1.1)

Step (3)

List of Drill checking

→ goto Machine → last option List of Drill (Now it will show list of drills)

diameter	No.
0.81	
1.02	
1.12	

• Not available 0.8 drill bit (holes) Resistor & capacitors ICs

• 0.81 → Any drill → Right click (select one drill)

(0.8) for circular soldering process

↓ edit all identical Pad

Pad apertures → change to 0.8

size 1.6 Drill 0.81

pad size 0.8 → double it → 1.6 size

Matched for checking or not matched size changed to 1.6.

goto Machine → List of Drill

• 1.12 → Any one drill → Right click

(1.1)

↓ edit all identical pad

Pad apertures

size 2.2 Drill 1.1 / 1.02

for Oval shape soldering

→ first x change to define as pad

1.02 Rightclick

(1) Some process like circular

1.02 → 1.00

Pad apertures

size 2 Drill 1

Step (4) Selected to

→ goto parameters →

step (5) List of d

→ goto Machine → list of dr

⇒ just for ma

step 6 Contour

→ goto machine → cont

step (7) G-Code

→ goto Machine → select Mill

Any circ

→ Now drill & Select is matched.

Step ② Allignment process

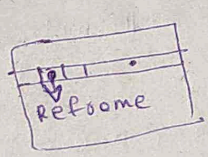
in this we should allign the circuit with the outline

→ Goto display → select Layer #6 . (Black colour line changed to some other colour)

→ goto file → offset → Co-ordinate offset

Shift manually.

no cursor will change to hand set. Now allign according to circuit.



click Reframe option . (Auto allignment)

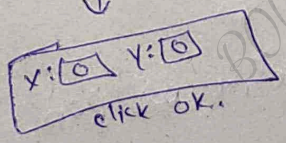
→ goto display → select layer 1 . (Now circuit will be selected)

fixed predefined layer { layer 5 → drill layer
layer 6 → cut layer

→ goto file → select flip horizontal (circuit changed to left to right (Mirror image)

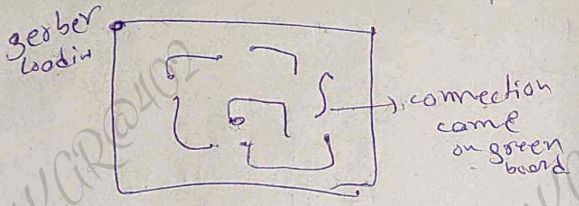
~~File origin~~

→ goto file → origin



Gerber loading
Step 1

1. Goto File → open (click)
   ↓
   • New circuit
   ↓
   choose your done gerber_file.
   • if not shown → click/select (All files)
   ↓
   • Select BOTTOM layer & open.
   gerber load close [X]
   pad aperture close [X]



• File → open → additional layer.
   ↓
   select 'cut' layer
   ↓
   open
   → dimension will be loading to select
   • give yes for foston tool

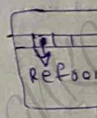
• File → open → Drill
   ↓
   select NCD file (open)
   ↓
   if incase Drill & circuit matches mean → Yes
   Not matched → NO
   ↓
   ⊙ trailing & predefined. (select) ⊙

→ Now drill & select

Step ② Alignment pro
in this we have to
outline

→ goto display → se

→ goto file → offset



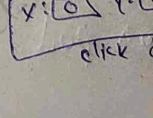
select

→ goto display → layer

fixed predefined layer { layer 5 → layer 6 →

→ goto file → select

→ goto file → origin



• Design completed

4. Software → Eagle - designing purpose
2. Copper CAM
3. Auto leveler
4. Mach-3 mill

• Why will we generate Gerber

Copper CAM (software purpose) → gerber file → converted to → G code (geometrical codes) (X, Y, Z-axis)

gerber file

Why converted ↓ bcoz machine understand G-code

Bottom
CUT
Drill

• Copper CAM (7 steps)

1. Gerber loading
2. Alligament process
3. List of Drill
4. Selected Tool.
5. List of Drill
6. Contours
7. G-code generation.

open copper CAM → green colour popup

Section [DRILL]

Device [EXCELLON]

File [%N.NCD]

layer → 44 Drills
45 Holes

DRILL	CUT	BOTTOM

→ give/click process Job.

clicking process Job

Before in the saved folder we have

• LCK File → 2 LCK file • B### File. • B#1 " • B#2 " • BRD " • S### " • S#1 " S#2 " S#3 " S#4 "	• SCH File • Text Document GPI

After clicking process Job

LCK File	NCD File
LCK "	S#1 File
B#1 "	S#2 File
BOTTOM "	SCH File
BRD "	Text Document
CUT "	
DRI "	
GPI "	

• Design completed

• Why will we generate

Copper CAM (software purpose) → gerber file

gerber file

Bottom
CUT
Drill

Why ce
↓
bcot

• copper CAM (7 S

1. Gerber load
2. Alligament pro
3. List of Drill
4. Selected T
5. List of T
6. Contours
7. G-Code ge

Open Copper CA

PCB → designing

→ need to give a titles.

11 items
↓
11 titles
↓

---

first unselect all the layers (Right side)

- section [*] delete "*"
  ↓
  BOTTOM
- Device · [GERBER_RS274X]
- File. [%N.BOTTOM]
  case sensitive
- Right side select layers → 16 Bottom
  → 17 Pads
  → 18 Vias

- click Add (to do further operations).

- section [CUT]
- File [%N.CUT]
- Layer → 20 Dimension
- Click Add

Step ③ (D) Routing

goto Edit → Collect Route

↓

Top ⟷ change to ⟷ Bottom

→ select [/] wise bend style 1

track should connect 45°

Not 90°

flow of electron will change suddenly means life time of track will be decreased

goto Edit → Rip up → to change or delete Routing line.

→ not use delete.

45° without Routing do not go to Gerber generation

→ Step ④

Geober Generation

goto file → CAM Processor

↓

one slide will open

PCB layout

1) Grid changes
2) Board dimension fixing
3) Compenents Placement
4) Routing <45° one tracks to one track some distance blw small distance components no track should be drawn

(2)(B) Goto → DRAW → Select dimension

16 Bottom → change to → 20 dimension

set 50

set 50 x 50 length of PCB board

Goto Edit → Select MOVE → change the outerline to 50x50.

by clicking at the center of the outerline you can change the outerline

goto Edit → Delete

delete the dimensions values showing (50) → click on 50 to delete

step (2)(B) Board dimensions fixed completed

step (3)(C) component placement

goto Edit → Select Move

Step-2 Bill of Material

open file "BR"

Goto File → Run ULP (select)

↓

Select bom.ulp

↓

click open

Bill of Material (opened)

↓

Save

↓

BR Save → close

• Saves at txt document

→ it contains all the details of components

Goto File → switch to board → Yes (click)

one pop-up slide will be opened.

Step-3

(3)(A)

View → Grid → Change inches to mm

size: 1 mm

Display ⊙on

Style ⊙Lines

click ok

PCB layout

1) Grid change
2) Board dimension Fix
3) Components Place
4) Routing (45° one to blw sm

(2)(B) Goto → Draw →

Set 50

Set 50 x 50

goto Edit →

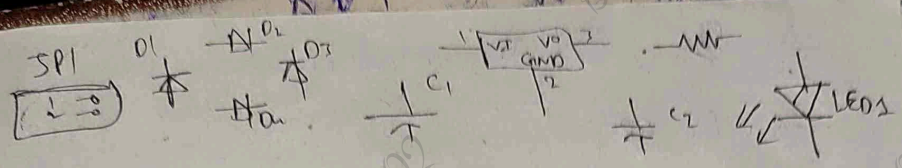
goto Edit → D

delete the dimen

Step-(3)(B) Board d completed

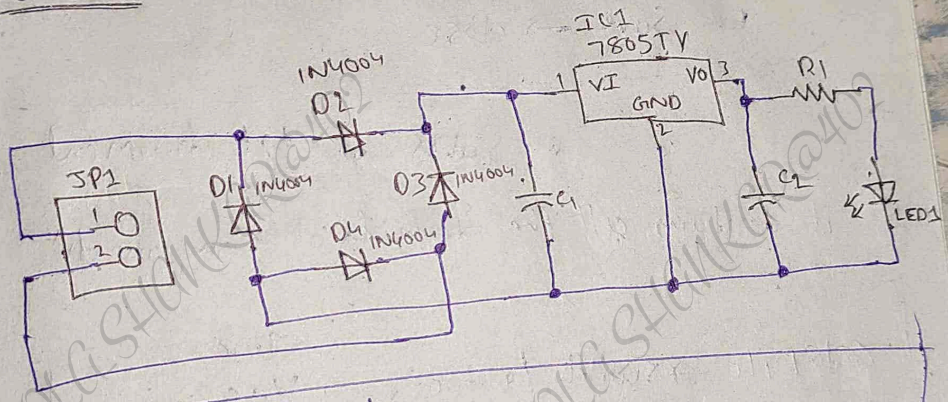
Step (3)(C) Com

goto Edit →



Goto - Draw → Net (choose)
. connect the circuit

Connections



Goto Edit → Select Value

C1 → 470UF
C2 → 0.1UF
R1 → 1K

CTRL + S → Save the file "BR".

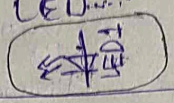
Edit → Add → Search (Led) (choosen)

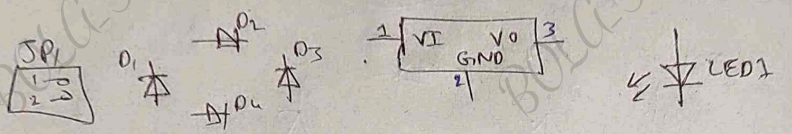
Selecting LED

~~→ LED 2AC = LEDS with 2 cathode & 2 Anod~~

→ LED (choose)

↳ LED... LED5MM





Edit → Add → ~~Search~~ (scl)

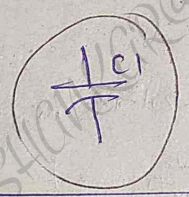
check "scl" in alphabetical order

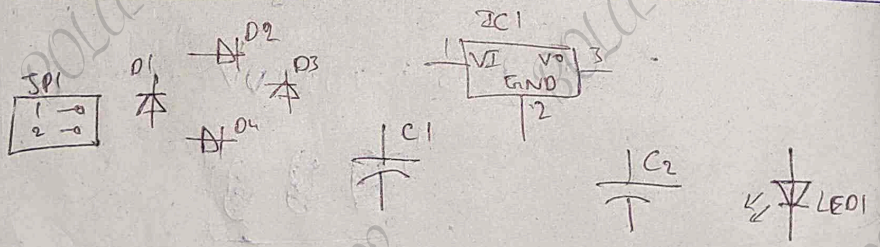
↓

C-US · capacitor, American Symbol (choosen)

↓

C-U... · C025-024X044 (choosen)





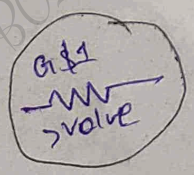
Edit → Add → · R(

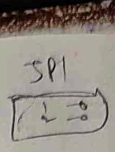
↓

R-US · Resistor, American Symbol

↓

R-U... 0207/10

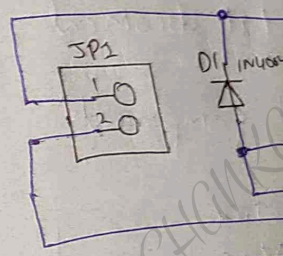




Goto - Draw → · Ne

· connect H

Connections



Goto Edit →

C1

C2

R

CTRL

Bridge rectifier used to blink one LED

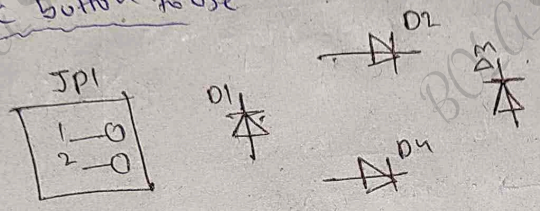
(9V) (step down transformer) → Resistor → (3.3V)

Edit → Add → Search. Diode

D1 IN4004

IN4004 (choosen)

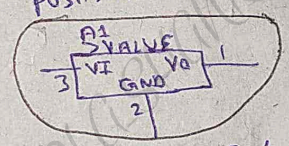
ESC button to use



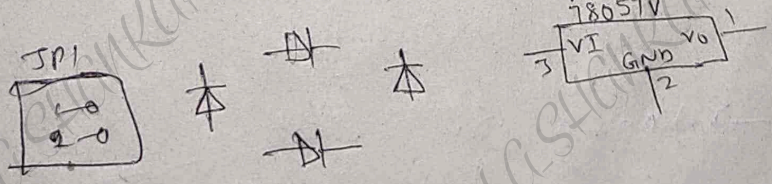
Edit → Add → Search. Regulator

(choosen)

→ 78* Positive voltage Regulator



→ 7805... TO22DV (choosen



EAGLE Software

4 steps

1. Schematic
2. Bill of Material (BOM)
3. PCB (board) x Layout design
4. Gerber generation

1. Schematic — step: (1)

- Goto File → New → schematic

  opens a schematic page

- we need to select grid option

- Goto View → Grid → Display ON

  Style ◉ Lines

  size : 1 (0.01 inches → MM) 2.54mm

  do not change values.

  click Ok

- Goto File → Save as "BR" name.

  D:\402\Bridge Rectifier

- Go to Edit → Add option select

  (this is the library page)

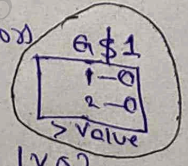
  Search [Pin header] click → Ok

  → Pin Header connectors

  (Red colour) Symbol → for schematic → Select 2 pin header

  footprint → for PCB layout → select Pin 1X02

  (Green colour)



Bridge rectifier Used (9V) step down transformer → Resist

Edit → Add

ESC button to use

JP1

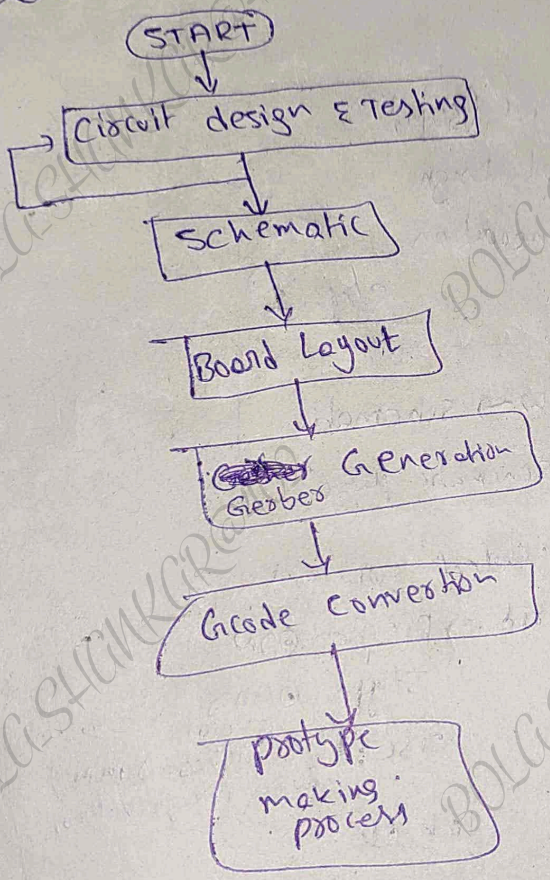
1
2

Edit → Add

JP1

1
2

PCB Design flow:



Electronic Design Automation (EDA)

EDA Tools – (PCB design softwares)

- Protel / Altium Designer
- Cadence – OrCAD, Allegro
- Mentor Graphics PADS
- Zuken CADSTAR

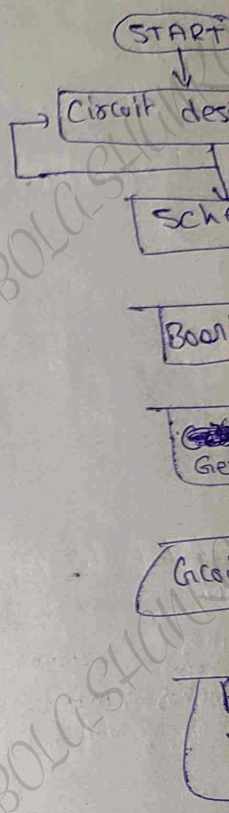
} licensed ver

- Public · Open source tools: cadsoft Eagle

proteus is also a PCB design software.
↳ simulation software
EAGLE → PCB software (not simulation).

PCB Design Flow

Nxt two days → EAGLE & PCB design softwares

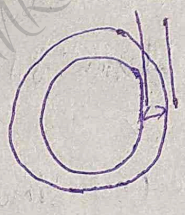


Arduino – blue colour solder masking
Raspberry pi – green colour

- Why colour different are used in PCBs.

→ Main reason to protect from oxidising
(main layer is copper layer)
: Colour depends on customer apperiens
Mostly green → track is very clear.
(after 1(or) 2 year very clearly visible for tracks)

Annular Ring:



Bare Board:

With components, the board is Bare Board.

Track:

track width → depends on power consumption or current consumption
depends on application – track width is designed.

Bill of Materials (BOM):

Types of PCB

- Single layer
- Double layer
- Multilayer (in layers) for connections
- Rigid flex PCB - laptop (keyboard to screen connections)
  - maintain the life
  - old nokia flip phone } examples
  - popup camera
- Aluminium PCB (observes heat)
  - Home appliances
  - LED Boards / LED panner

component placement
Track connections

increase the life

PCB Terminology

FR-4
- copper thickness - 0.035 mm (top layer)

FR-4 copper plate

PCB Materials

types
- FR1

Soilder Masking

Arduino - blue . colour
Raspberry pi - green c

- Why . colour .

→ main reason to p
(main layer is
: Colour . depends on
Mostly green

Annular Ring:

Bare Board:
with compone

Track:
track with
depends on

Bill of M

onents
& Passive)
e active / voltage)
ource / voltage
external source voltage or current)

size of PCB ↓
complexity ↓
cost ↓

Passive component SMT

Passive two terminal components.

Package Type	Size in inches	Size in mm
0201	0.024" x 0.012"	0.6mm x 0.3mm
0402	0.04" x 0.02"	1.0mm x 0.5
0603	0.063" x 0.031"	1.6 x 0.8
0805	0.08" x 0.05"	2.0 x 1.25
1206	0.126" x 0.063"	3.2 x 1.6
1210	0.12" x 0.10"	3.2 x 2.6
2020	0.20" x 0.20"	5.08 x 5.08
2512	0.25" x 0.12"	6.35 x 3.0

Transistor & Diode Package

1. SOT-23 small outline transistor
2. SOD-123 - Small outline Diode.

Integrated circuit SMD Package

pin to pin distance

- Small outline Integrated circuit (SOIC) - 1.27mm
- TSOP (Thin Small outline Package)
- QFP (Quad Flat Pack)
- QFN (Quad Flat Non-leaded Pack)
- PLCC (plastic leaded chip carrier)
- PGA (Pin Grid Array)
- BGA (Ball Grid Array)

DIP
1 pin to another pin distance = 2.54mm
in through hole or breadboard

RHAGAVAN

Electronic components

→ Basics of components (Active & Passive).
→ Types of components.

(consume/produce some external voltage).

Active - depending external source/voltage

Passive - not depending (not consider any external source voltage or current).

Mounting Technologies

1. Through Hole technology
2. SMT technology.

→ size of PCB ↓
complexity ↓
cost ↓

Passive component SM

Passive two terminal c

Package Type	size
0201	0.024" x
0402	0.04" x
0603	0.063" x
0805	0.08" x
1206	0.126" x
1210	0.12" x
2020	0.20" x
2512	0.25" x

Transistor & Di

1. SOT-23 Sma
2. SOD-123 -

Integrated circu

- Smad outline I
- TSOP (Thin small o
- QFP (Quad flat
- QFN (Quad Flat
- PLCC (plastic le
- PGA (Pin Gr
- BGA (Ball

DIP

1 pin to another pin dista
in through hole @ bre

PC

what is PCB ?

its importance ?

PCB designing (next 3 days).

Agenda

- Basic of electronic component type.