

Vistas 2D de PCB

Diferentes Capas (LAYERS)

Prof. Andrés Roldán Aranda

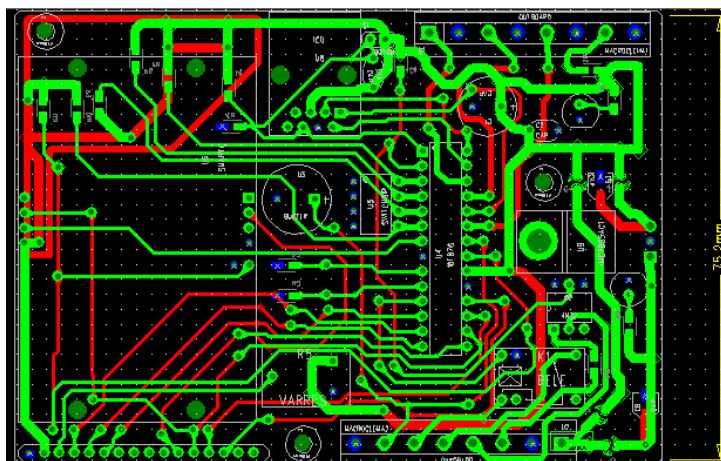


Curso
PCB 2006



Universidad de Granada
Departamento de Electrónica y Tecnología
de Computadores

Ejemplo Simple a Doble Cara

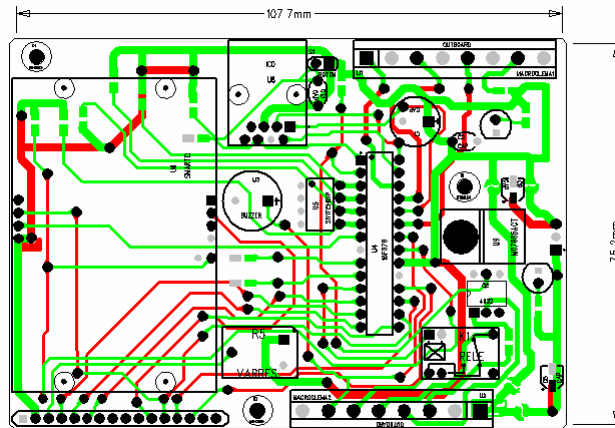


Curso
PCB 2006

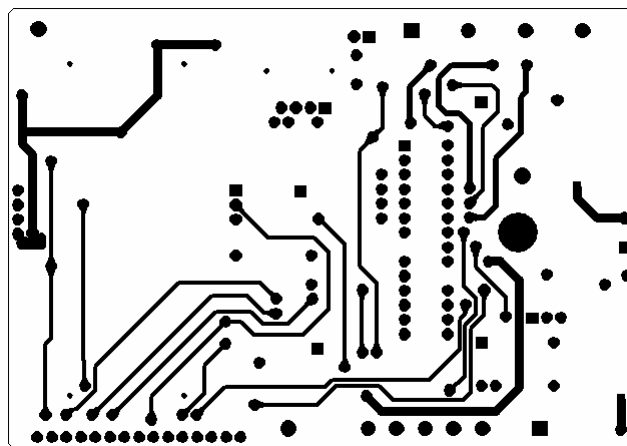


Universidad de Granada
Departamento de Electrónica y Tecnología
de Computadores

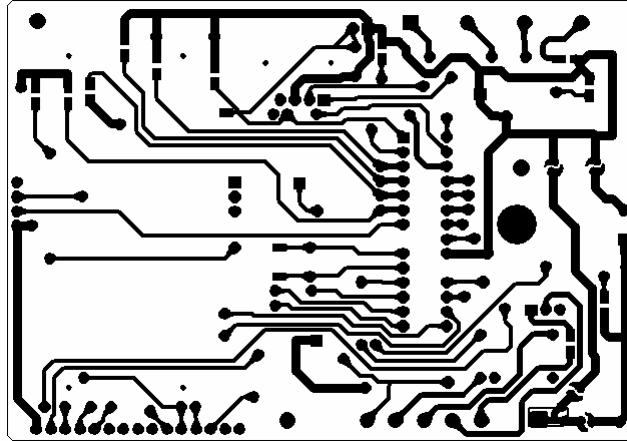
Top + Bottom + Acotaciones



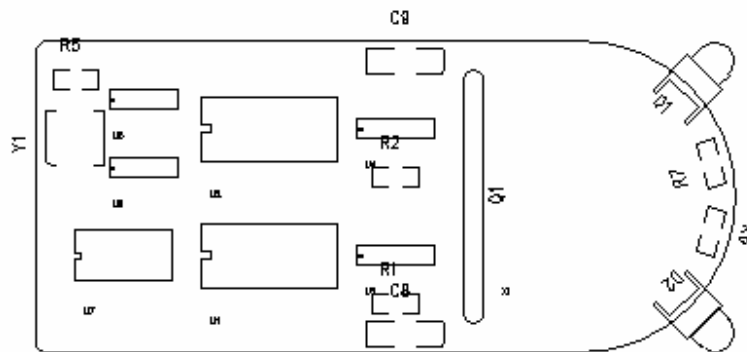
TOP



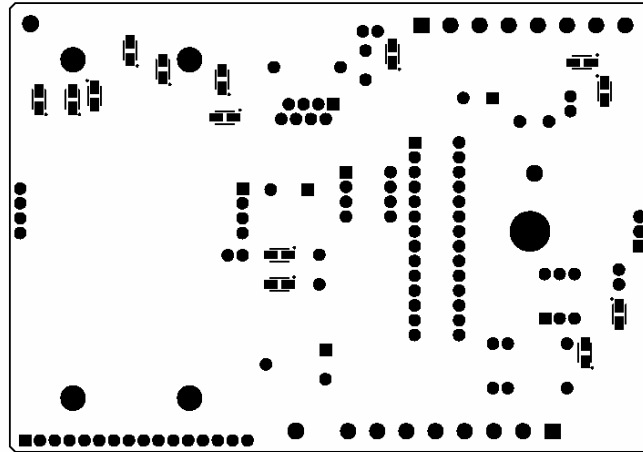
Bottom



Silkscreen Top



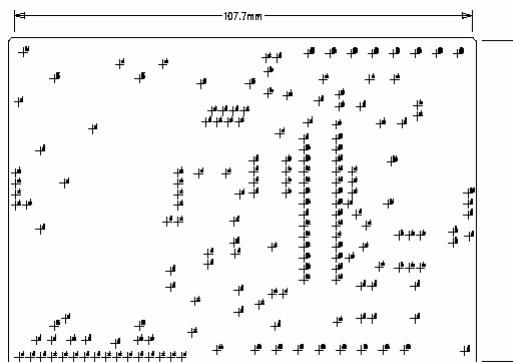
Silkscreen Bottom



¿Sobra algo?

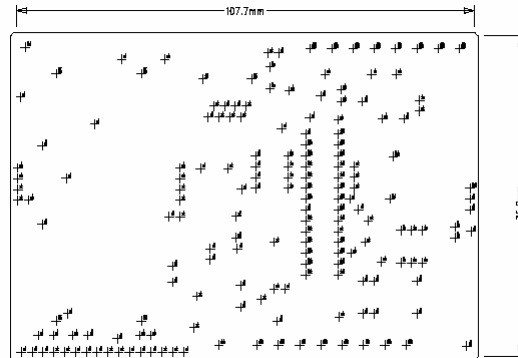
TOP Drilling Chart

SIZE	QTY	SYM	PLATED	TOL
1	112	+	YES	+/-0.0
2	6	+	NO	+/-0.0
1	15	+	YES	+/-0.0
1	26	+	NO	+/-0.0
2	4	+	YES	+/-0.0
1.5	16	+	YES	+/-0.0

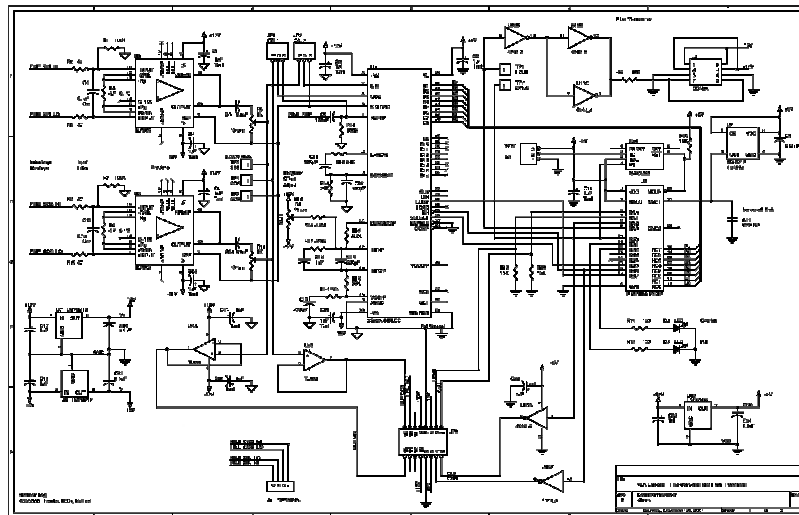


BOTTOM Drilling Chart

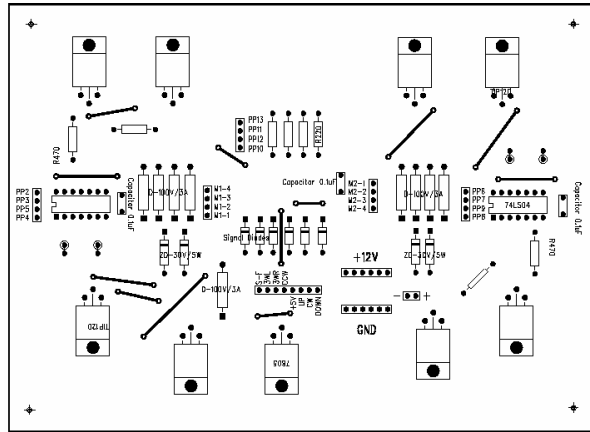
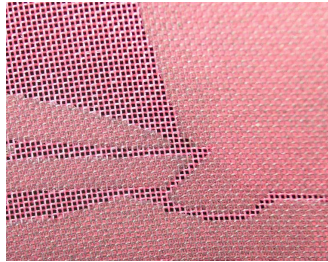
SIZE	QTY	SYM	PLATED	TOL
1	112	⊕	YES	+/-0.0
2	6	⊕	NO	+/-0.0
1	15	⊕	YES	+/-0.0
1	26	⊕	NO	+/-0.0
2	4	⊕	YES	+/-0.0
1.5	16	⊕	YES	+/-0.0



Esquema



SILKScreen – Serigrafía {TOP, BOTTOM}

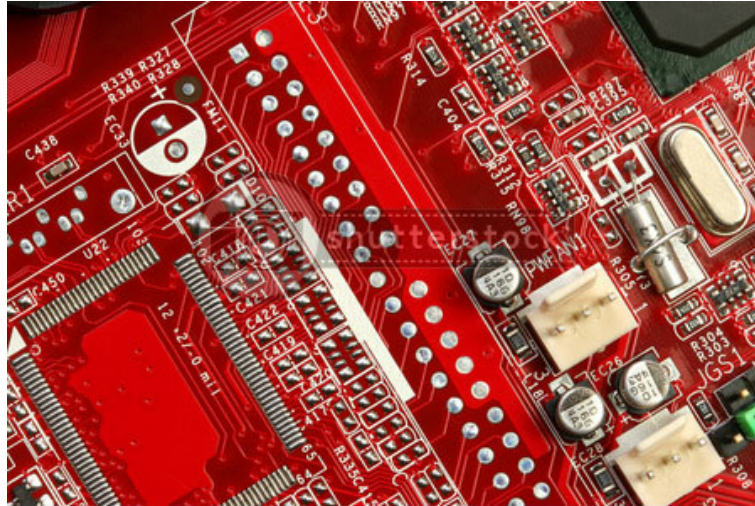


Drill Chart

TOOL	CODE	SYMBOL	
T1		+	
T2		^	
T3		□	
T5		△	

TOOL	CODE	DIMEN	DIAM	ED AN	PLATE P	TELETYPE
T1	1	1.0	0.35	0.0	NO	0.35/0.35/0.35
T2	2	1.0	0.50	0.0	NO	0.50/0.50/0.50
T3	3	1.0	1.00	0.0	NO	1.00/1.00/1.00
T4	4	1.0	1.50	0.0	NO	1.50/1.50/1.50
T5	5	1.0	2.00	0.0	NO	2.00/2.00/2.00
T6	6	1.0	2.50	0.0	NO	2.50/2.50/2.50
T7	7	1.0	3.00	0.0	NO	3.00/3.00/3.00
T8	8	1.0	3.50	0.0	NO	3.50/3.50/3.50
T9	9	1.0	4.00	0.0	NO	4.00/4.00/4.00
T10	10	1.0	4.50	0.0	NO	4.50/4.50/4.50
T11	11	1.0	5.00	0.0	NO	5.00/5.00/5.00
T12	12	1.0	5.50	0.0	NO	5.50/5.50/5.50
T13	13	1.0	6.00	0.0	NO	6.00/6.00/6.00

SILKScreen – Serigrafía CONDENSADOR



www.shutterstock.com · 1113809

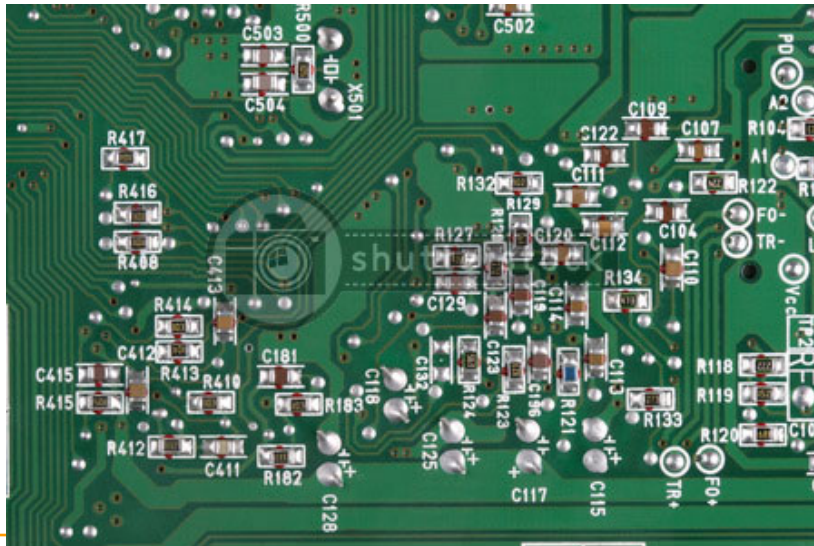


Curso
PCB 2006



Universidad de Granada
Departamento de Electrónica y Tecnología
de Computadores

SILKScreen – Serigrafía debe LEERSE BIEN



www.shutterstock.com · 5529893



Curso
PCB 2006



Universidad de Granada
Departamento de Electrónica y Tecnología
de Computadores

1a

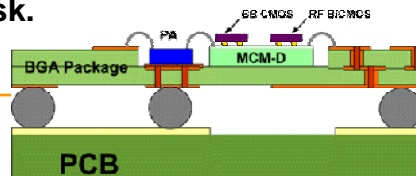
DES/SOLDAR máquinas de vacío



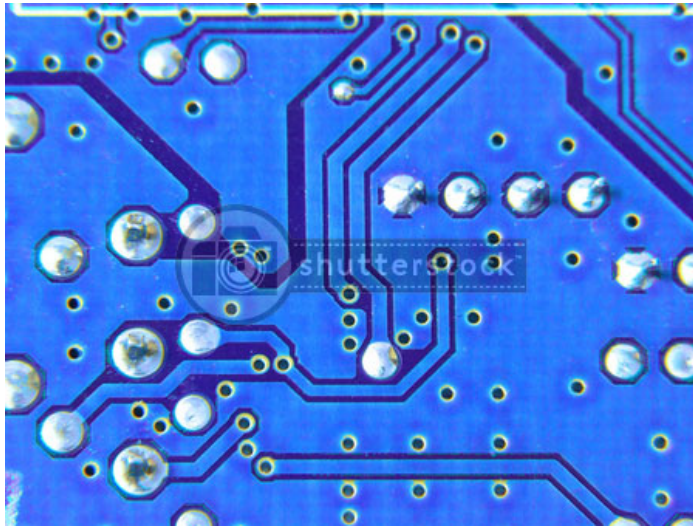
www.shutterstock.com · 7242235

MÁSCARA de SOLDADURA: Estaño no agarra

- **Solder mask** is usually the **green coating** on a PCB board which is designed to insulate and protect the underlying copper traces from environmental factors, and is also used to prevent bridging (shorting) traces during wave soldering
- **Solder mask** usually covers everything on the PCB board except for pads and vias, though it is good practice to cover vias, especially if dealing with BGA components. This process is called *tenting* the vias
- **Solder mask** is shown on the CAD tool as a negative image. I.e. where there is solder mask “shown” is where there will be **NO** soldermask.



MÁSCARA de SOLDADURA: Estaño no agarra

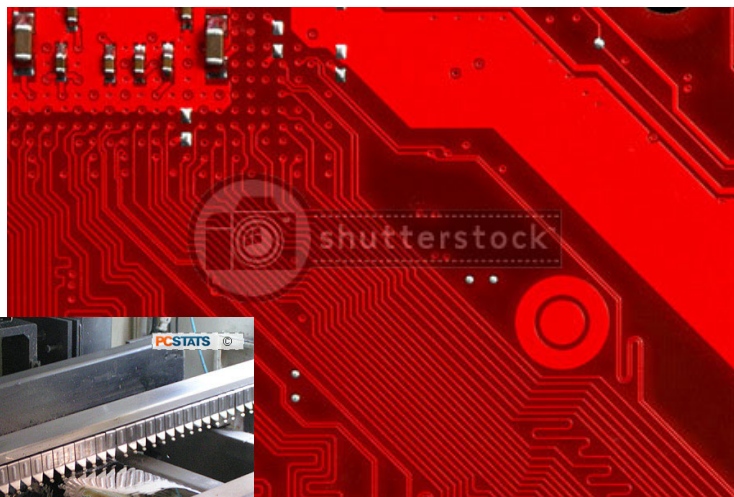


Curso
PCB 2006



Universidad de Granada
Departamento de Electrónica y Tecnología
de Computadores

Solder Mask - Protección frente a cortocircuitos en soldadura por ola de estaño



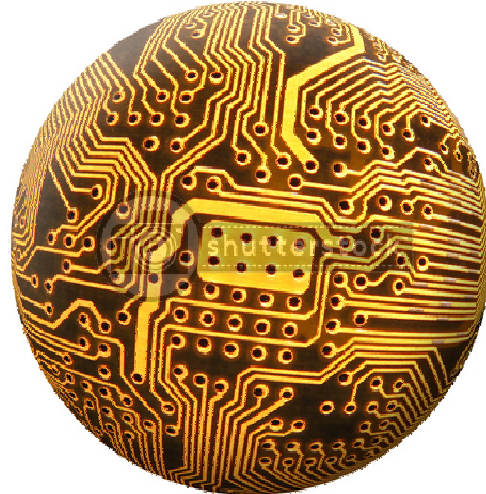
shutterstock.com · 1113819

Curso
PCB 2006



Universidad de Granada
Departamento de Electrónica y Tecnología
de Computadores

VIAS entre diferentes CAPAS



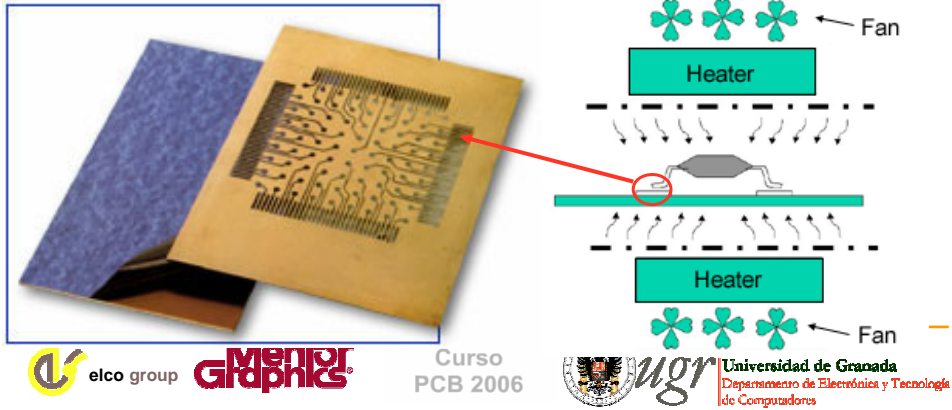
SOLDER Mask protege el Cu de Oxidación



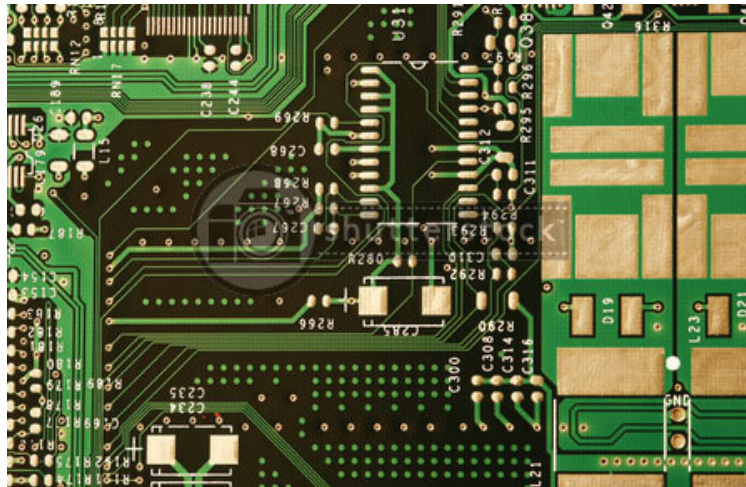
PASTE MASK

Paste Mask

- Paste mask is similar to solder mask, except that it is used to create solder paste screens which can then be used to solder SMDs in a hot re-flow soldering process.

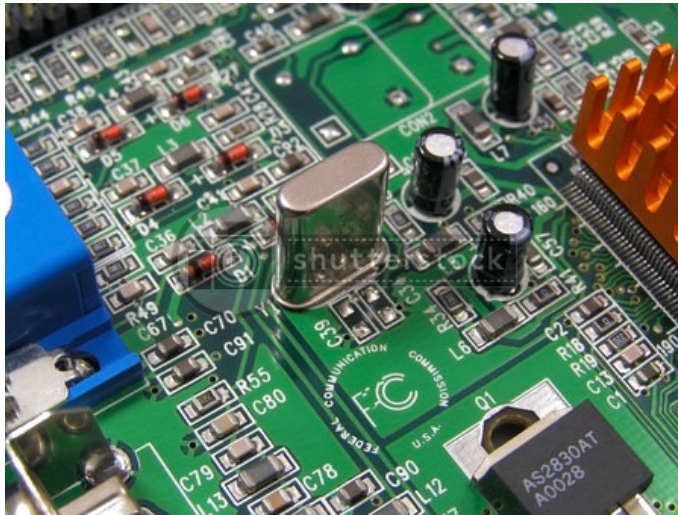


Prestañado de PADS, Pasivado PADS

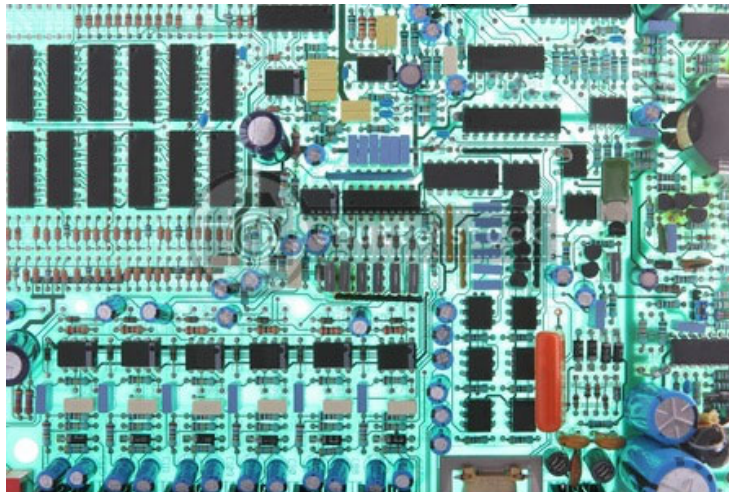


www.shutterstock.com · 914448

Serigrafía de IMAGENES (Autocad)

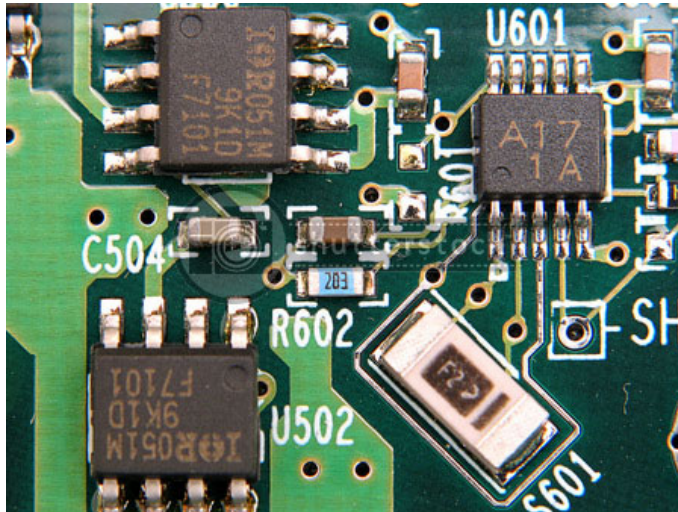


Orientación de los componentes



www.shutterstock.com · 1547914

FANOUT para poder salir de CAPA



www.shutterstock.com · 171579



Curso
PCB 2006



Universidad de Granada
Departamento de Electrónica y Tecnología
de Computadores

Zonas de Reserva para PULSADORES Baño de Oro



www.shutterstock.com · 752479



Curso
PCB 2006

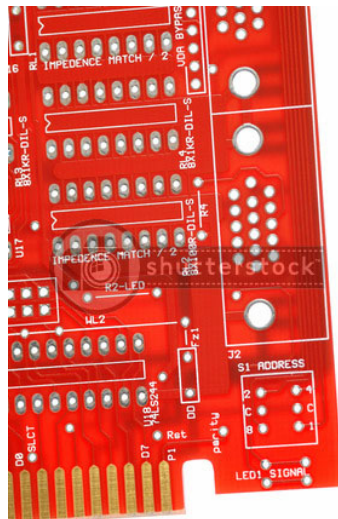


Universidad de Granada
Departamento de Electrónica y Tecnología
de Computadores

SILKSCREEN - Serigrafía

- The silkscreen layer is also known as Overlay.
 - **Top Overlay** refers to the silkscreen on top of the board, and **Bottom Overlay** refers to silkscreen on bottom.
 - This is the layer onto which the component designators are printed (R1, R2, ...) so as to identify individual components during component placement of the board.
 - They are also used during the PCB routing process to indicate the outlines of your components. This helps you in placing (or not placing) components too close to one another, or too close to the edge of the board.
- **Make sure your silkscreen doesn't run over any exposed copper (such as pads)!**

Pines ESTAÑADOS, Conector PASIVADO



Agujeros en la PCB



www.shutterstock.com · 752484



elco group



PCB 2006



Granada
Departamento de Electrónica y Tecnología
de Computadores

VIAS

- Vias are special pads which connect electrical signals from one side of your board to another.
- In special circumstances, from one layer to another without crossing all layers (**blind or buried vias**)
- **Blind and buried vias** are to be avoided at all cost, they are difficult to debug and rework.
- Vias are made of conductive material which are called **Plated Through-hole**.
- There is really no difference between Vias and Pads except that the CAD tools manage them differently to allow more complex operations on vias.
- Vias are generally much smaller than pads



elco group

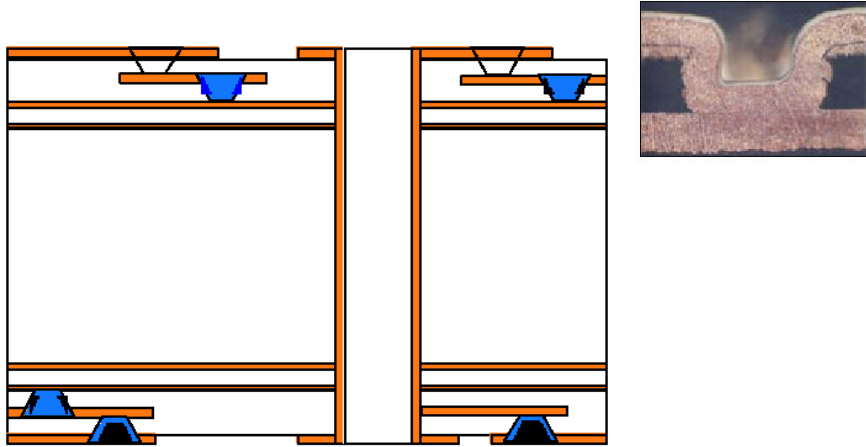


Curso
PCB 2006

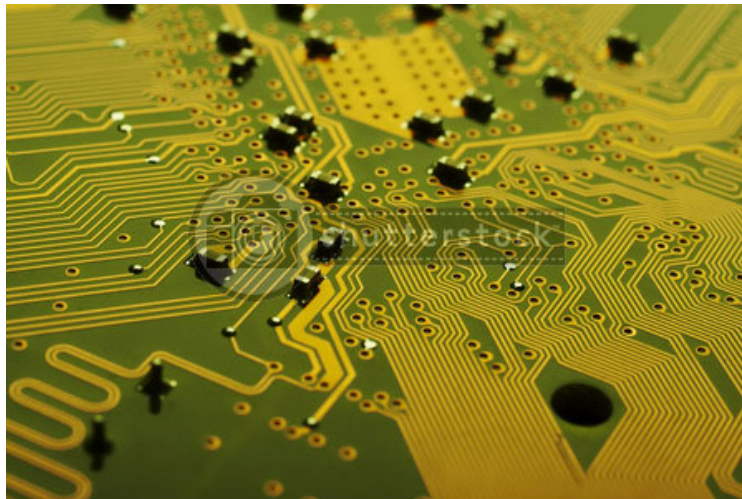


Universidad de Granada
Departamento de Electrónica y Tecnología
de Computadores

VIAS



Differential NETs



www.shutterstock.com · 1389114

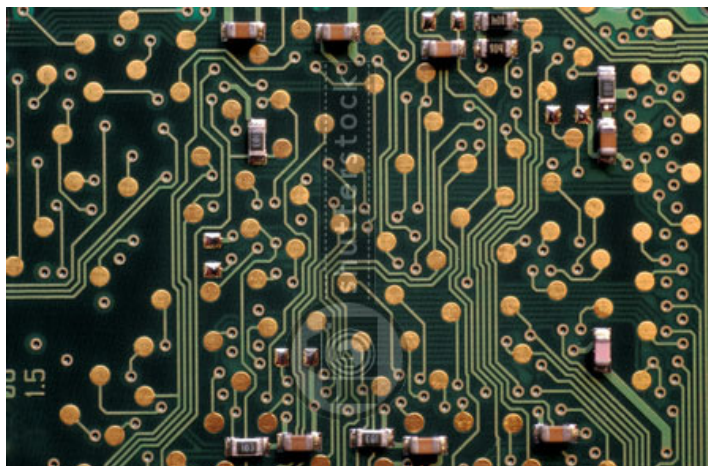
UNIDADES MÉTRICAS

- PCB Boards are primarily designed in **Imperial units** (inches [“]) as opposed to metric units (mm).
- A thousands of an inch is called **mil** (not to be confused with mm), where:

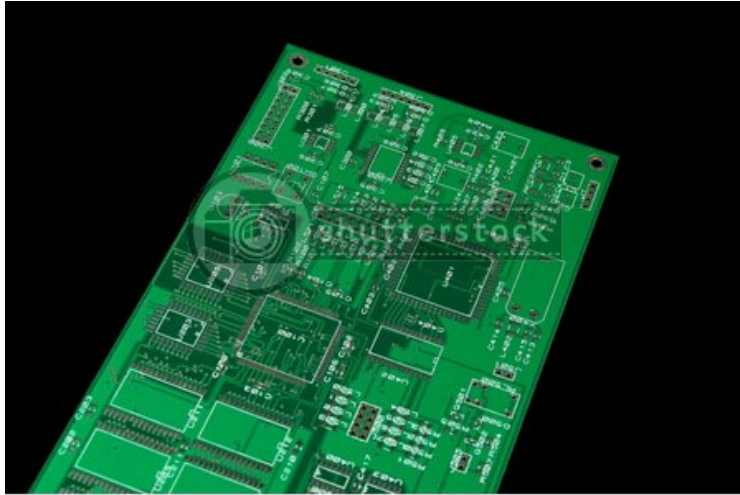
$$100 \text{ mils} = 0.1 \text{ inch} = 2.54 \text{ mm}$$

- The reason for using imperial units in a PCB document is because most of the components were manufactured according to imperial pin spacing. The practice continues even today!

TEST POINTS



Agujeros de Soporte.



www.shutterstock.com · 1040966

Chip on PCB



www.shutterstock.com · 5178604

GROUNDING & Bypassing

- Providing good grounding is critical to the functionality of a PCB board
- On a multi layer board, one of the layers should be dedicated ground and all ground signals should have vias into this ground plane.
- **Avoid chaining grounds** in order to prevent ground loops from occurring.
- Use **bypass capacitors** to smooth out power spikes by components that suddenly draw significant current Bypass caps should be placed as close to the component's power pins as possible. You should use **100 nf caps as the norm**, with lower capacitance for higher frequencies, and higher capacitance for lower frequencies



Curso
PCB 2006



Universidad de Granada
Departamento de Electrónica y Tecnología
de Computadores

VIA PATTERNS minimizar resistencia



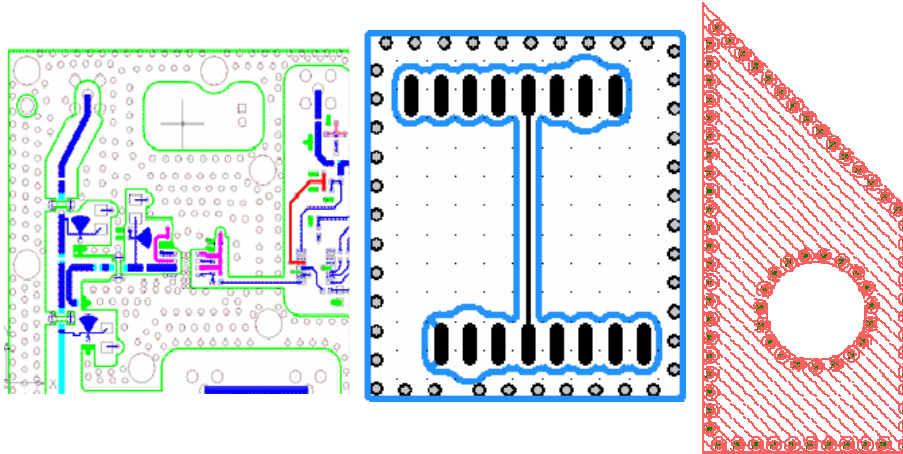
www.shutterstock.com · 935189

PCB 2006



Universidad de Granada
Departamento de Electrónica y Tecnología
de Computadores

VIA PATTERNS minimizar resistencia PADS



Tecnología de Montaje THT - Through Hole Tecnolgy



www.shutterstock.com - 4127086

Esnablaje completo de la PCB

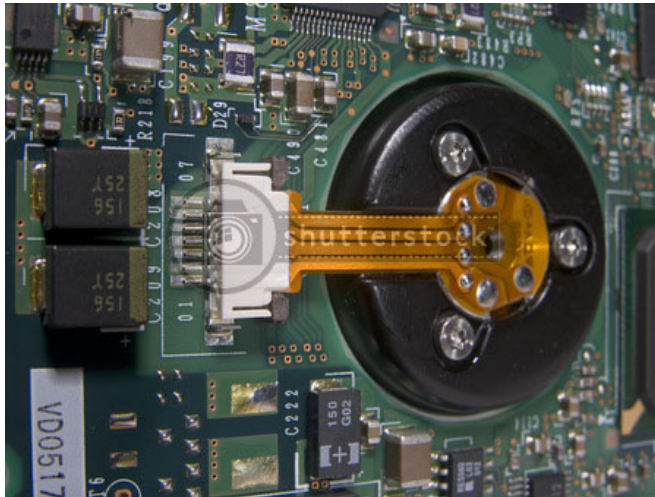


Montaje de Soportes metálicos con tornillería.



www.shutterstock.com · 1581029

CONECTORES en SMD



www.shutterstock.com · 5178613

PCB Simple Cara – THT



www.shutterstock.com · 711254

INTERRUPTORES y JUMPERS



www.shutterstock.com - 1551445

Coexistencia de Tecnologías THT - SMT

