Embedded Systems

Achitectures and Design Styles
(part 1 – PCB based design)

Prof. William Fornaciari
Politecnico di Milano, DEIB
William.fornaciari@polimi.it

Outline

- **Architectures and approaches to the design**
  - **Printed Circuit Board**
    - **Components**
    - **Support**
    - **Mounting**
    - **Design**
  - System-on-Chip
    - Design
      - Design for testability
  - Distributed Embedded Systems
    - Application fields
    - Wireless sensor networks
    - Design
  - Platforms for prototyping
    - Types of systems
    - Design

Architectures and approaches to the design

Architecture and design

- Select the architecture is the most important step
  - Quality and cost are affected significantly, even the feasibility
  - Phases of the development, in general
    - Collection of specifications and requirements
    - Definition of the system architecture (Hw, or Hw/Sw)
    - Partitioning of Hw, Sw and communication
    - Implementation of the Hw, Sw and communication sections
    - Production
  - Depending on the type and complexity of the system, different development models can be considered (e.g., waterfall, V-shape, Spiral)

Architecture and design

- Well known problem to face with
  - The cost of redesign increases during the development of the product, dramatically
  - Sometime the choice is driven by the attitude to the risk
- The intrinsic heterogeneity of the embedded systems and the broad range of constraints, makes hard identifying a «good» architecture
  - There is no standard methodologies to this purpose, frequently the previous experience is the only valid option
    - Empirical methods, quantitative evaluation, similarity with other projects, virtual prototyping, …

Hardware Architectures

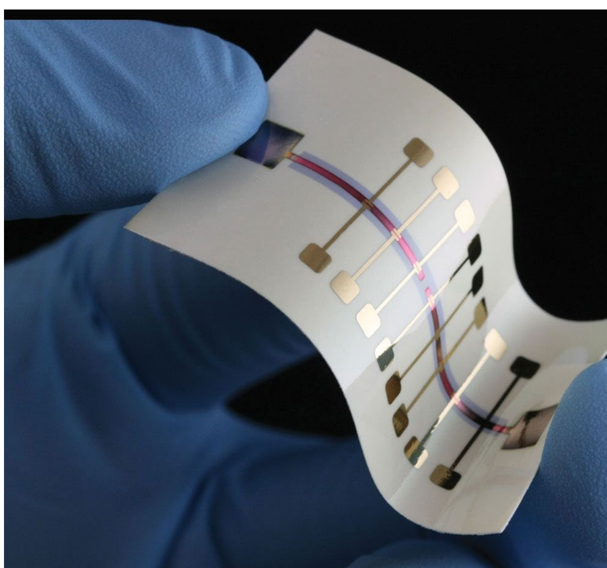
– Printed Circuit Boards (PCB)

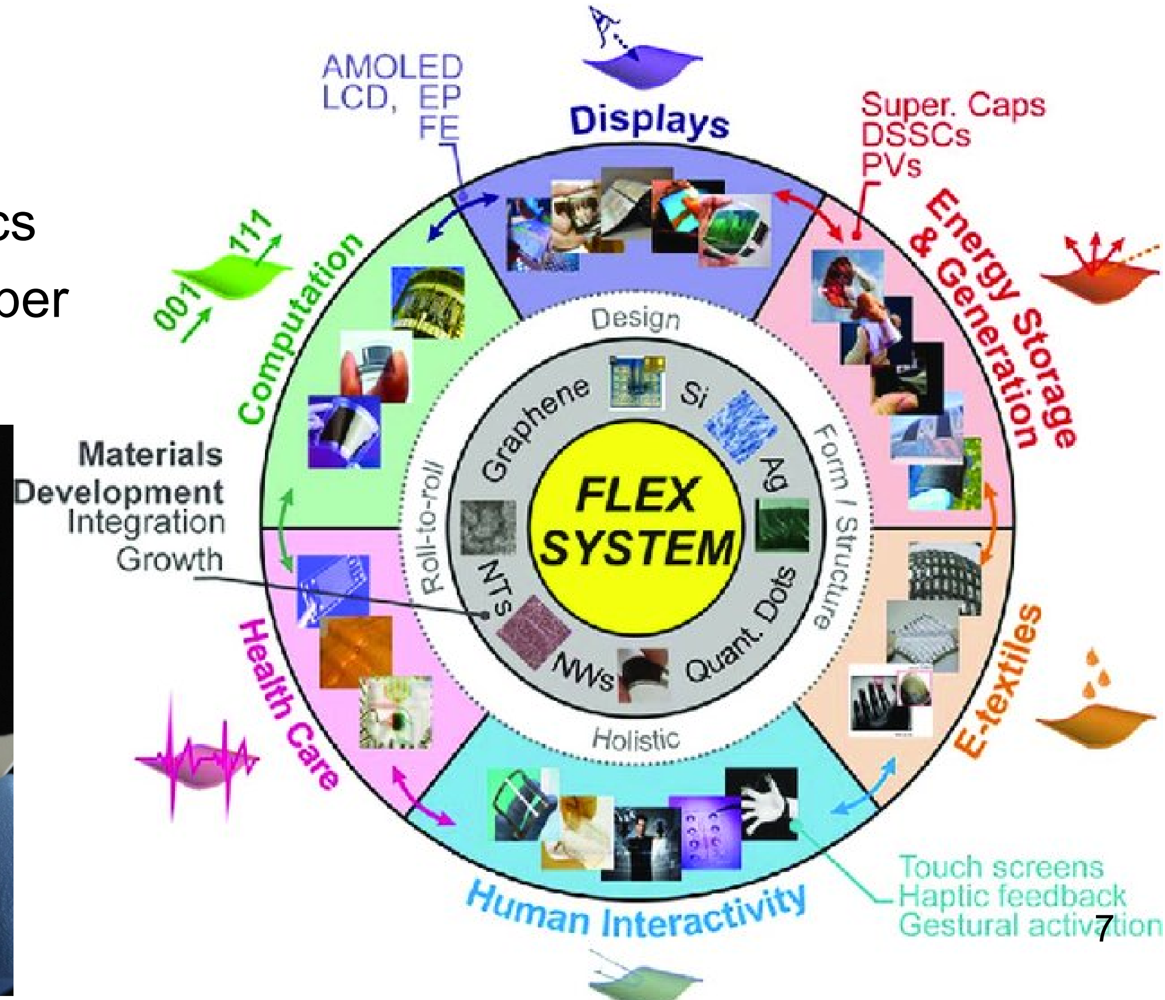
– System-on-chip (SoC)

– Distributed Embedded Systems

Printed Circuit Board

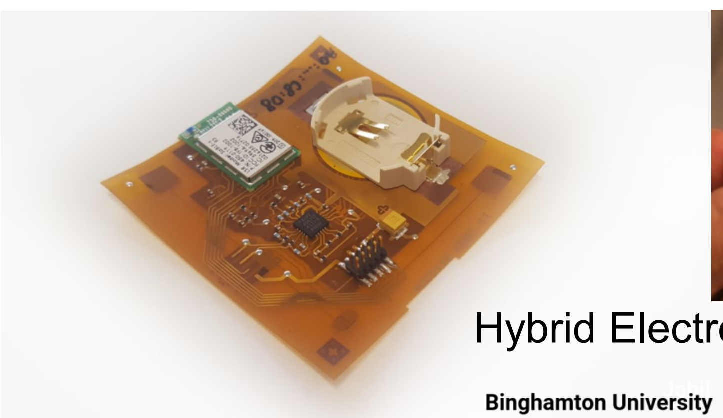
- Components with small form factor assembled on a plastic support
- New technologies
  - Flexible electronics
  - Electronics on paper

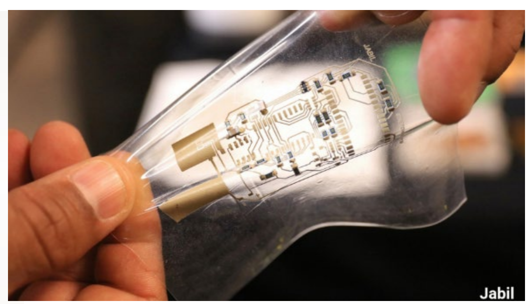




Flexible electronics

- **Flexible Electronics**: generally refers to a class of electronic devices built on conformable or stretchable substrates, usually plastic, but also metal foil, paper and flex glass
- **Flexible Hybrid Electronics (FHE)**: printed electronics combined with silicon-based integrated (active) circuits on a conformable substrate. Examples of key active components from Si CMOS processes include microcontrollers, digital signal processors, high density memories and RF chips





Hybrid Electronics

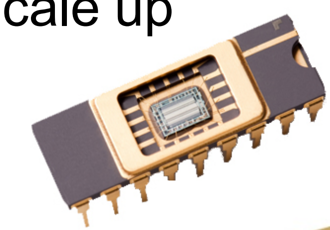
Flexible Battery

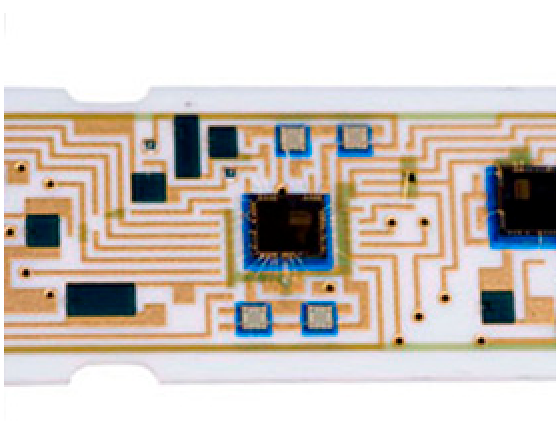
Flxible electronics

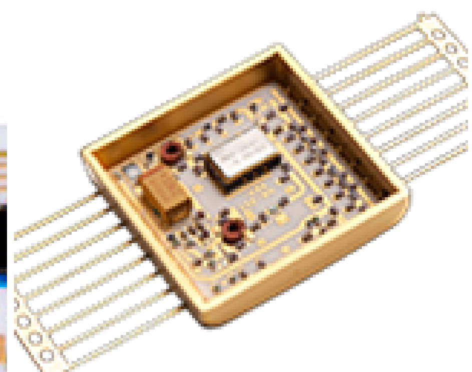
- **Organic Electronics**: a field of material science concerning the design, synthesis, characterization, and application of organic small molecules or polymers that show desirable electronic properties such as conductivity. Some freely substitute this term with plastic electronics
- **Plastic Electronics**: generally refers to a class of electronic devices built on plastic (polymer) substrates, as opposed to silicon or glass
- **Printed Electronics**: functional electronics fabricated by laying conductive lines using one of several printing methods, including: screen, ink jet, gravure, flexography and others. Often confused with printed circuit boards, which also often use printing methods to connect discrete active and passive components

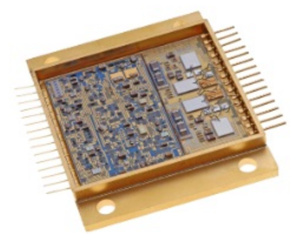
System-on-chip (SoC)

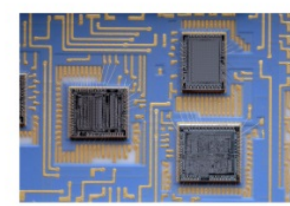
- The entire systems integrated in a single chip
  - Something remains outside mounted in a small PCB
    - Connectors, antennas, slots for flash memory, …mounted on a small PCB
  - The design differs from that of the PCB, since the integration scale offer more possibilities to scale up
    - SoC
    - MPSoC
    - MCM and Hybrid microcircuits
    - NoC





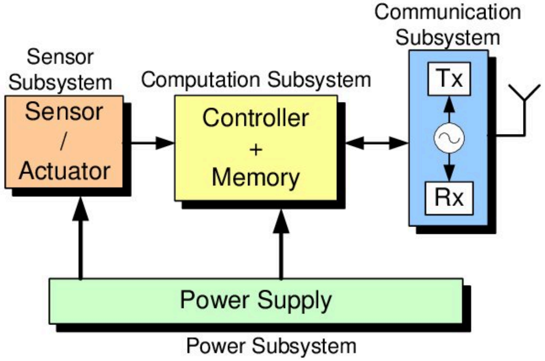


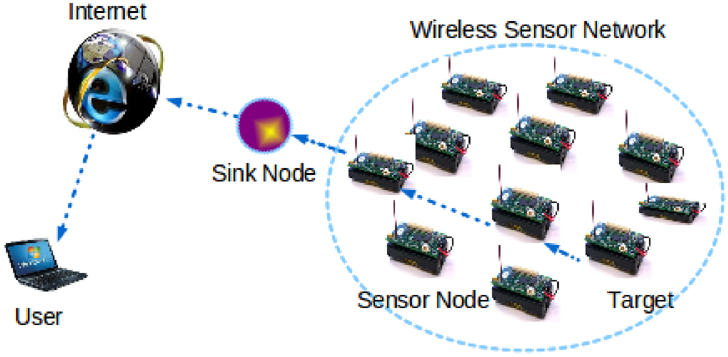




Networked Embedded Systems

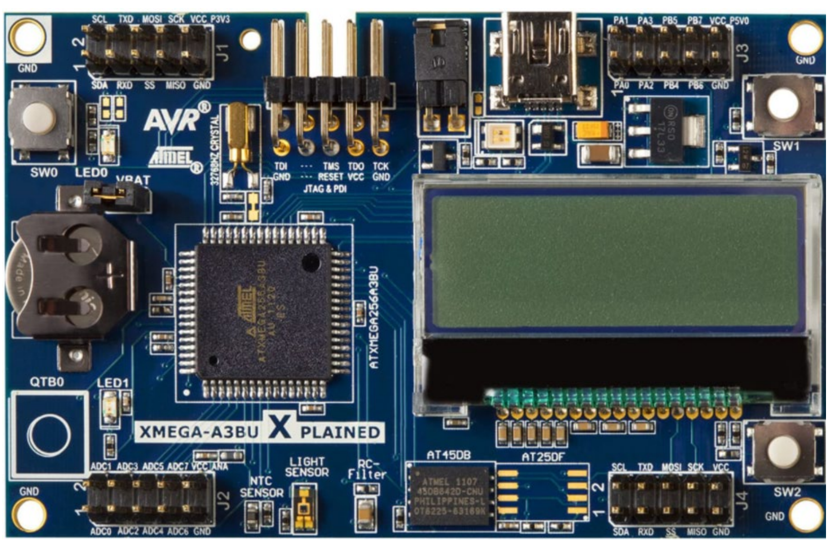
- Interactive systems with wired or wireless connections
  - Notable example: WSNs
    - Tiny sensor nodes, with limited computing capabilities and wireless connection
      - Extreme case: near zero computing
    - Automotive: basically wired connectivity among the native functions of the car. Entertainment could be wireless
    - Domotics: wired+wireless, telemedicine, smart home
      - Long life of devices, cost of the infrastructure





Architecture and design

- In any case, note that:
  - Prototyping/evaluation boards are crucial to develop embedded systems of a certain complexity
  - Frequently they integrate FPGAs, DSPs, or both and several peripherals.
    - Fast development of complex and complete applications
    - Mitigation of the problems related to integration and interfacing; focus of the effort on the application (if the software in bundle is

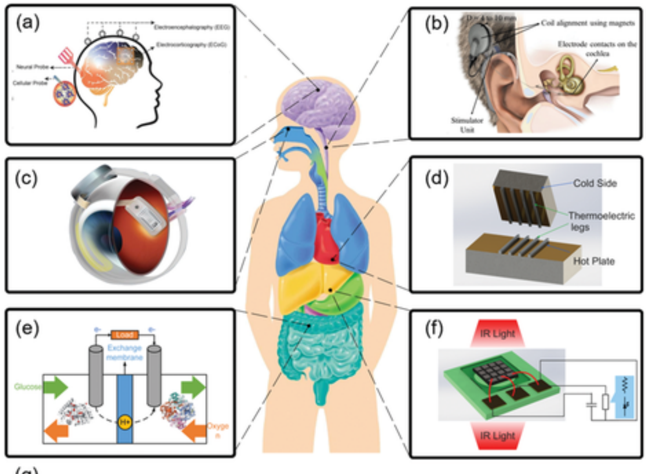


Example (around 50 €):
Xplained Microchip, CPU AVR
ATXMEGAA3BU-XPLD
•Microcontroller ATxmega256A3BU
•QTouch® button
•Analog sensors
•3 mechanical buttons
•2 LEDs
•4 Expansion pins

Printed Circuit Board

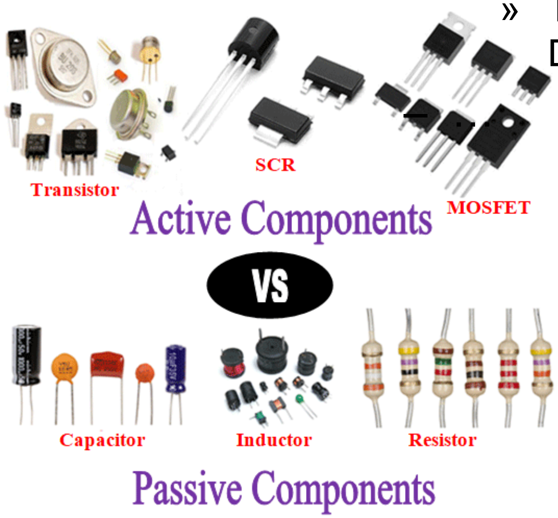
Printed Circuit Board

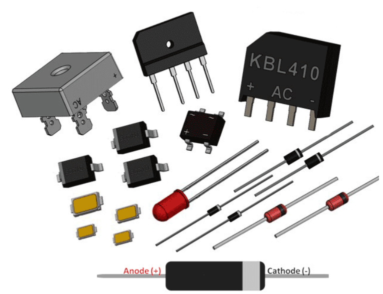
- **PCB-based** systems employ off-the-shelf components (**COTS**), with different types of packages, or tailored for specific application classes
  - COTS – based design
    - Term frequently used when the system is largely using COTS mounted on a PCB
    - Each component is typically associated to basic functionalities
- Usually, any embedded systems is using -even a small- PCB
  - Only when the form factor is critical other solutions could be adopted

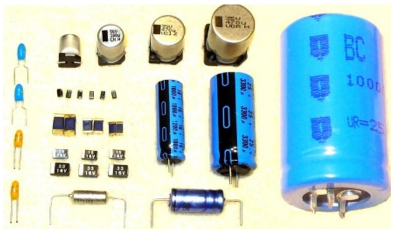


Printed Circuit Board

- Components
  - The first step of a PCB-based design is the choice of the COTS
    - The most common types of components are
      - Passive
        - » Connectors, socket, dip-switch, buttons, LED, resistors, capacitors, coils,…
      - Power Supply
        - » Each subsystems must the properly powered, a specific power distribution must be created
      - Converters and filters
        - » To interface the «analog» external world, it is necessary to use filters, ADC, DAC, amplifiers, oscillators and, sometime, power stages

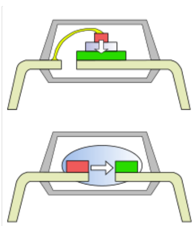






Printed Circuit Board

- Components



  - Electro-optical components
    - » To interface electro-magnetic signals from optical signals, e.g. opto-isolators, foto-diodes, laser diodes, infrared sensors
  - RF components
    - » Antennas, amplifiers, mod/demod stages

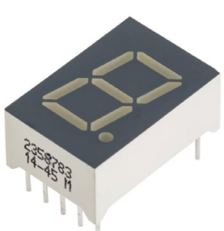
Filter

Coupler

Power divider

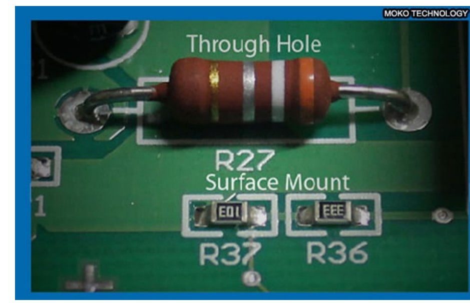
Isolator / circulator

  - Display
    - » 7-digit display, LCD display, alphanumeric display, LCD B/W or color, touch-screen, …
  - Sensors
    - » Conversion in Voltage/Current (or resistance) of physical parameters
  - Digital components
    - » In most of the ESs, the data processing is digital (Hw or Sw)
    - » Wide range of components, from dedicated ICs, programmable logic, microprocessors, microcontrollers, etc

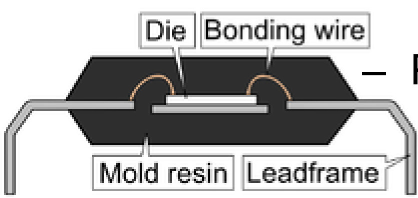


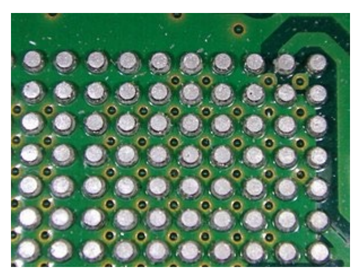
Printed Circuit Board

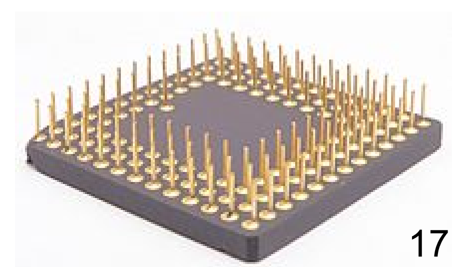
- Components and packaging
  - Frequently, there are alternative packages for a component, with different levels of protections, size and cost
    - Main characteristics of a package
      - Mounting
        - » Through-Hole (TH), Surface Mounted (SMD)
        - » Mixed solutions are possible
      - Positioning of the pins
        - *In-Line*: SIP, DIP, PDIP; *Small-Outline*: SOJ, SOP, TSOP
        - *Quad Surface Mount* (CC, QFP); *Grid Array* (PGA, BGA)
      - Material of the package
        - » Most of them are plastic, or ceramic, sometime metal (at a cost!!!)



DIP







Why through-hole is still there?

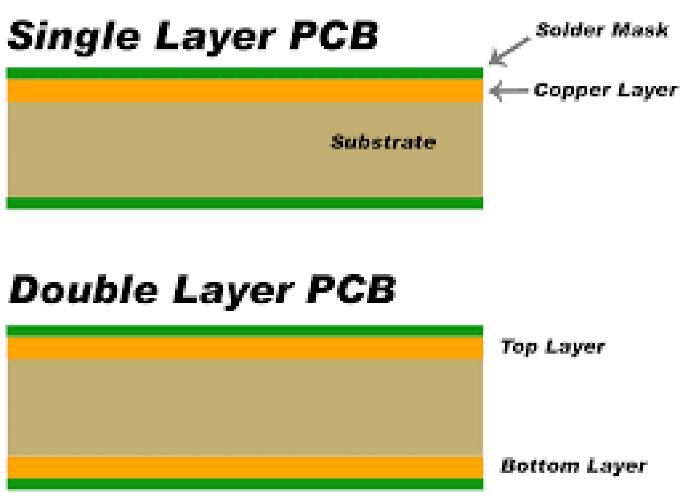
- Through-hole connection to the PCB is mechanically stronger than most surface-mount connections
- Through-hole components are best used for high reliability
  - While surface-mount devices (SMDs) are secured only by solder on the surface of the board, through-hole component leads run through the board, allowing the components to withstand more environmental stress
  - Through-hole is commonly used in military, automobile, and aerospace products that may experience extreme accelerations, collisions, or high temperatures
- Manual prototype assembly or component-level repair is more difficult for SMDs. SMDs cannot be used directly with breadboards
- SMD solder connections may be damaged by potting compounds going through thermal cycling
- SMDs are unsuitable for large, high power, or high voltage parts, such as in power circuitry

Printed Circuit Board

- Components
  - Depending on the positioning of the components and of the mounting techniques and number of pin, there are packages with different characteristics
    - Main parameters to be considered
      - Effectiveness
        - » Ratio between the area of the *die* and that of the package
      - Pin-count
        - » Number of pins (8-2000)
      - Pitch
        - » Space between pins (2.5-0.6 mm)
      - Thermal conductivity (resistance)
        - » Capability to transfer the generated heath from the *die* to the package (°C/W)

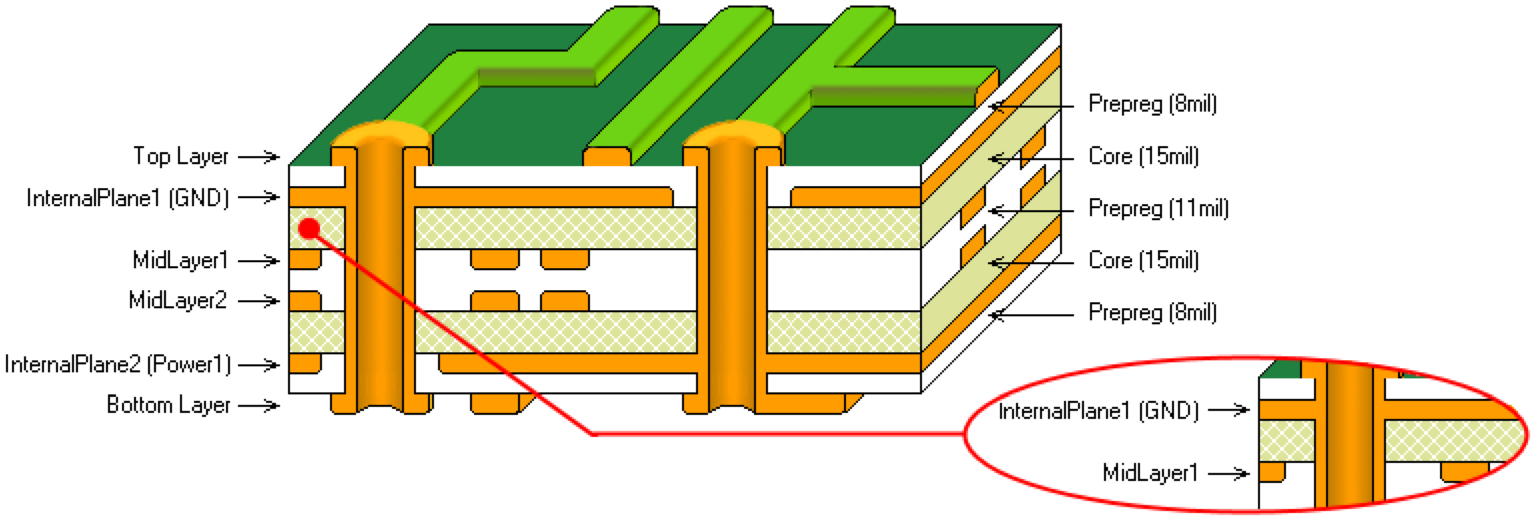
Printed Circuit Board

- Physical support
  - Consist of three main components
    - Conductor
    - Insulator
    - Glue (still insulator)



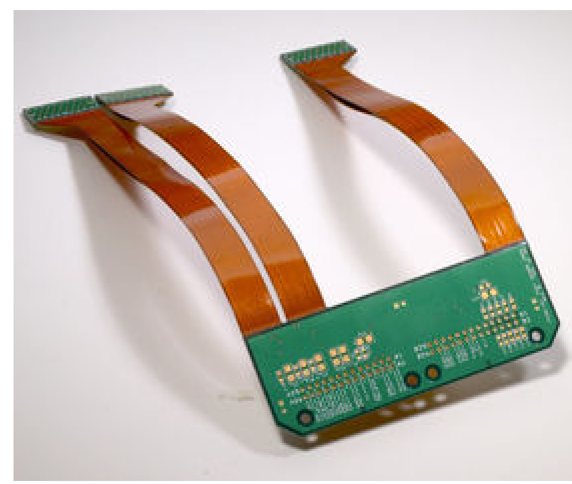
Printed Circuit Board

- Support
  - Conductive layer
    - Copper
    - Use for wires and interconnection, ground planes, contact pads where to weld the components
    - In multi-layer boards, typically 4-8 layers for supply/ground and 6-10 for wires



Printed Circuit Board

- Support
  - Insulator
    - Double goal
      - to prevent electrical short circuits among wires
      - to provide mechanical support for mounting of the components
    - Most used material: Fiberglass Epoxy Resins, FR-2 and FR-4
    - In some cases, it can be used Kevlar or Kapton to realize flexible supports



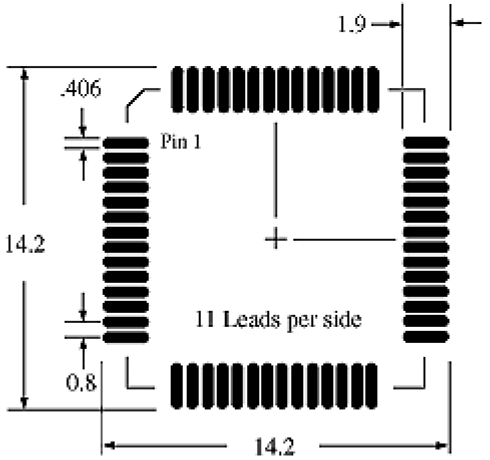
Printed Circuit Board

- Support
  - Adhesive (glue)
    - Dielectric layers containing the wirings, are joined using adhesive materials (fiberglass impregnated of epoxy resin) called *pre-peg*
      - Pre-preg layers, at room temperature, are not adhesive solid foils. At high temperature and pressure, they become more fluid and adhesive, while maintaining the property to be insulators
      - Going back to room temperature, at normal pressure, the epoxy resin become solid again, with the desired mechanical, chemical and electrical properties

Printed Circuit Board

- A board is a stack of layers of insulator, where their faces host the wires, separated by layers of pre-peg.
- The industrial process to realize a PCB is quite complex and encompasses a sequence of phases
- The starting point is a file (Gerber, Excellon, ecc…) that is the output of the design using a CAD program
- Three versions of Gerber formats are available:
  - Gerber X2 - the newest Gerber format with stackup data and attributes contained
  - RS-274-X - an expanded version of Gerber format and it has been widely applied
  - RS-274-D - the oldest version of Gerber format which is being gradually replaced by RS-274-X

Footprints



- Design libraries are available for most parts
- New footprints can be added manually
- Often footprints can be downloaded from the part vendor or from Topline (http://www.toplinedummy.com)
- There are IPC design guidelines (IPC-SM-782 at http://www.ipc.org) and Jedec component definitions (http://www.jedec.org)
- In prototypes, you're most concerned with fitting the part on the board properly, but in real products we consider joint geometry for manufacturing yield and product reliability
- (footprint = pad dimensions and land patterns)

Gerber

- Gerber format file describes design requirement of each image of a circuit board and it can be applied for both **bare board fabrication** and PCB **assembly**
  - When it comes to **bare board fabrication**, Gerber format is called for by both standard photoplotters and other manufacturing equipment desiring image data like legend printers, direct imagers or AOI (Automated/Automatic Optical Inspection) equipment etc. put it simply, Gerber format files have to be depended from beginning to the end of PCB fabrication process
  - When it comes to **PCB assembly**, a *stencil* layer is included in Gerber format and component locations are regulated as well, which will be regarded as significant reference data for SMT (Surface Mount Technology) assembly, thru-hole assembly and mix of them

Information to provide when ordering PCBs

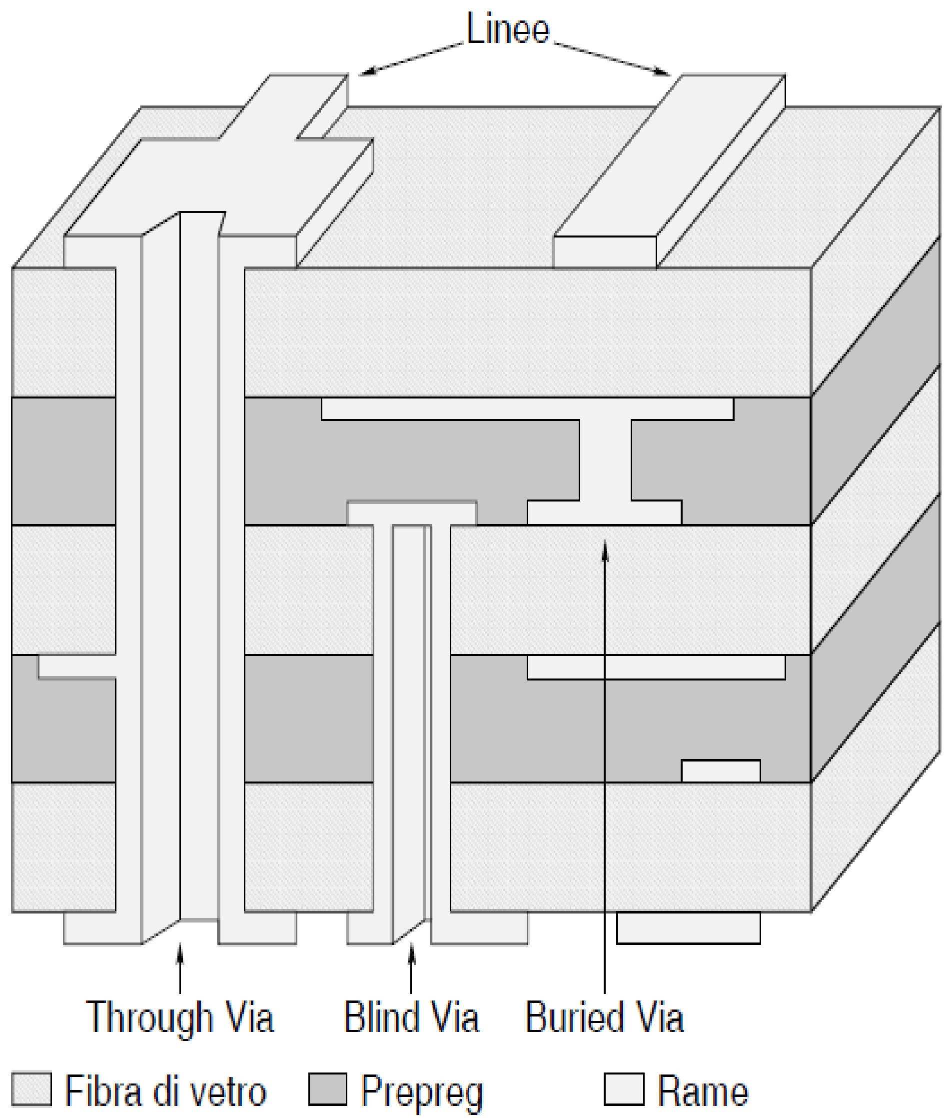
1. Quantity and lead time
2. X-Y dimensions/boards per panel, number of sides with components
3. Board material, thickness (4 layer boards usually 0.062”) and tolerances
4. Layer count and copper weight for layers:

   -½ oz or 1oz copper on outer layers (less copper means shorter etch times)
   - 1 oz copper on inner layers (carry more current for ground/power planes)
5. Metallization (SnPb/HASL, organic, Cu-Ni-Au, immersion Sn or Ag or Au)
6. Minimum line and space width (< 0.008” costs more)
7. Hole count, min hole dim and finish (holes < 0.015” cost more)
8. Surface mount pad count and minimum pad pitch
9. Silkscreen and solder mask (usually green LPI)
10. Electrical testing requirements (need netlist for electrical test)
11. Gerber data (always create a README file)

Printed Circuit Board

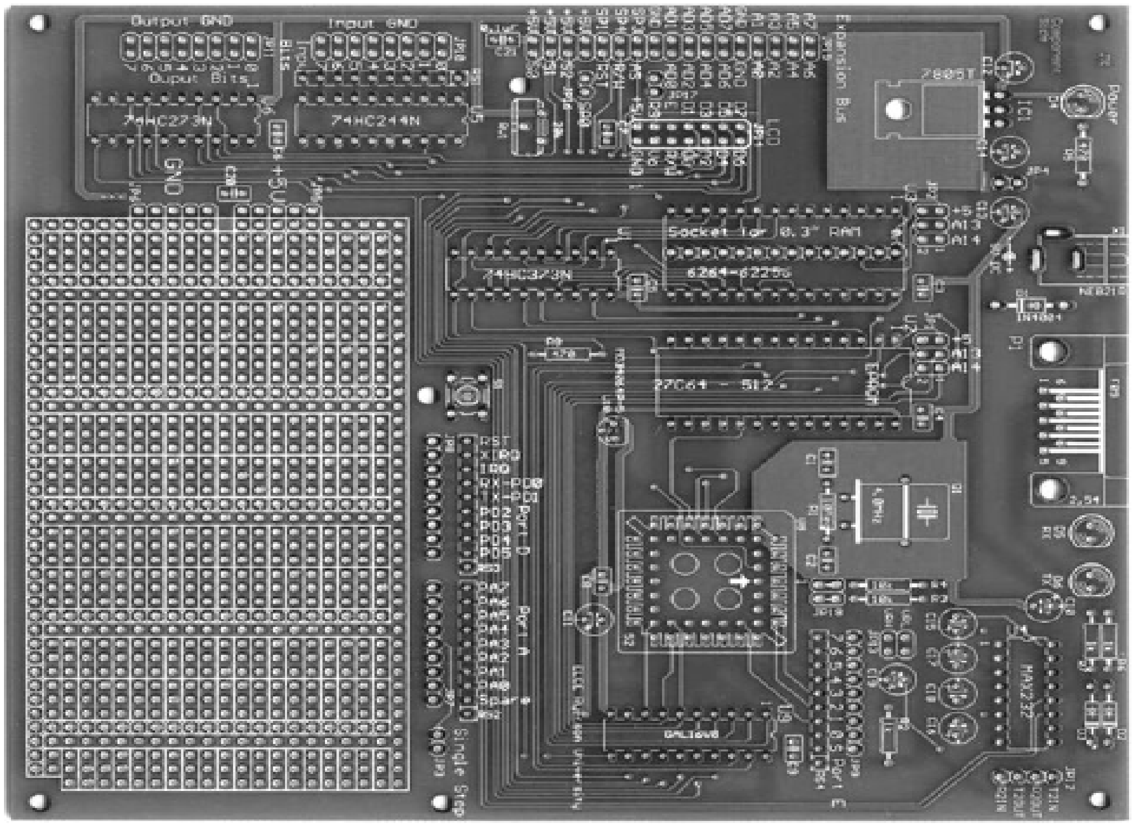
- Steps to realize a PCB
  - Realization of the dielectric supports
  - Realization of the wires
    - It is used a foto-sensitive resin (*photoresist)* that, when exposed to the light, become resistant to acids. In such a way, the not protected areas could be removed, exposing and creating the wires
  - Assembly of layers
    - The payers created by means of a photolithographic process and stacked and alternated to pre-peg foils. A heated press finalizes this step
  - Drilling (*via*)
  - Fabrication of the external layers
  - Finishing

Printed Circuit Board



Printed Circuit Board

- Final result



Printed Circuit Board

- After the realization of the board it is necessary an electrical testing (*bare-board testing*) to verify if the contact points ad connected or isolated
  - Simplest way: check the resistance among a pair of points
    - bed-of-nails
    - *flying-probe*
    - X-Ray inspection
      - *Image recognition*

- Board-testing is a slow, costly and crucial step
  - Frequently part of the (non functional) design is devoted to support debug and testing

Printed Circuit Board

- Mounting
  - For small boards and limited volumes (e.g., prototypes), the mounting is carried out manually
  - For volume and/or high complexity, specific equipment is used
    - Usually who develop PCBs is only in charge of the design, the fabrication of the board and the mounting is outsourced
      - Manual positioning is critical in terms of accuracy
        - » For packages like SIP or DIP, the pitch allows a manual welding.
        - » For QFP, PGA or BGA, manual operation is impossible

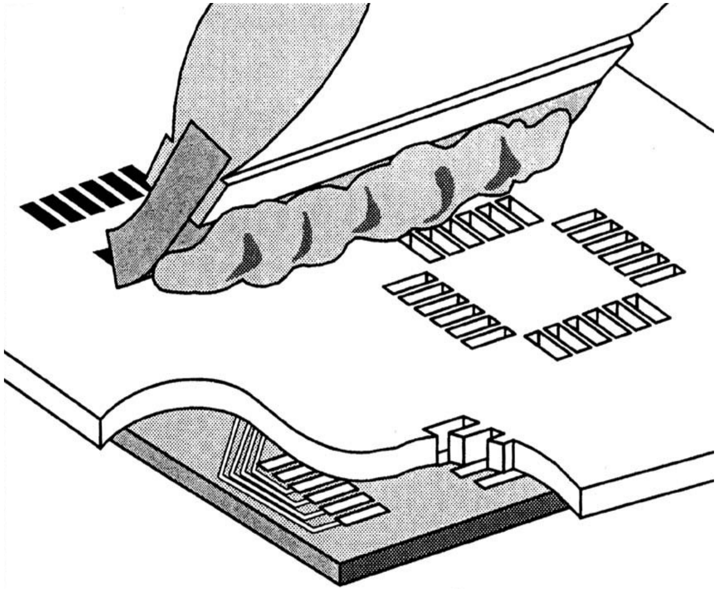
Assembly

Surface mount assembly process steps

- Solder paste printing or dispensing
- Component placement
- Reflow
- Inspection
- Rework/backload
- Cleaning

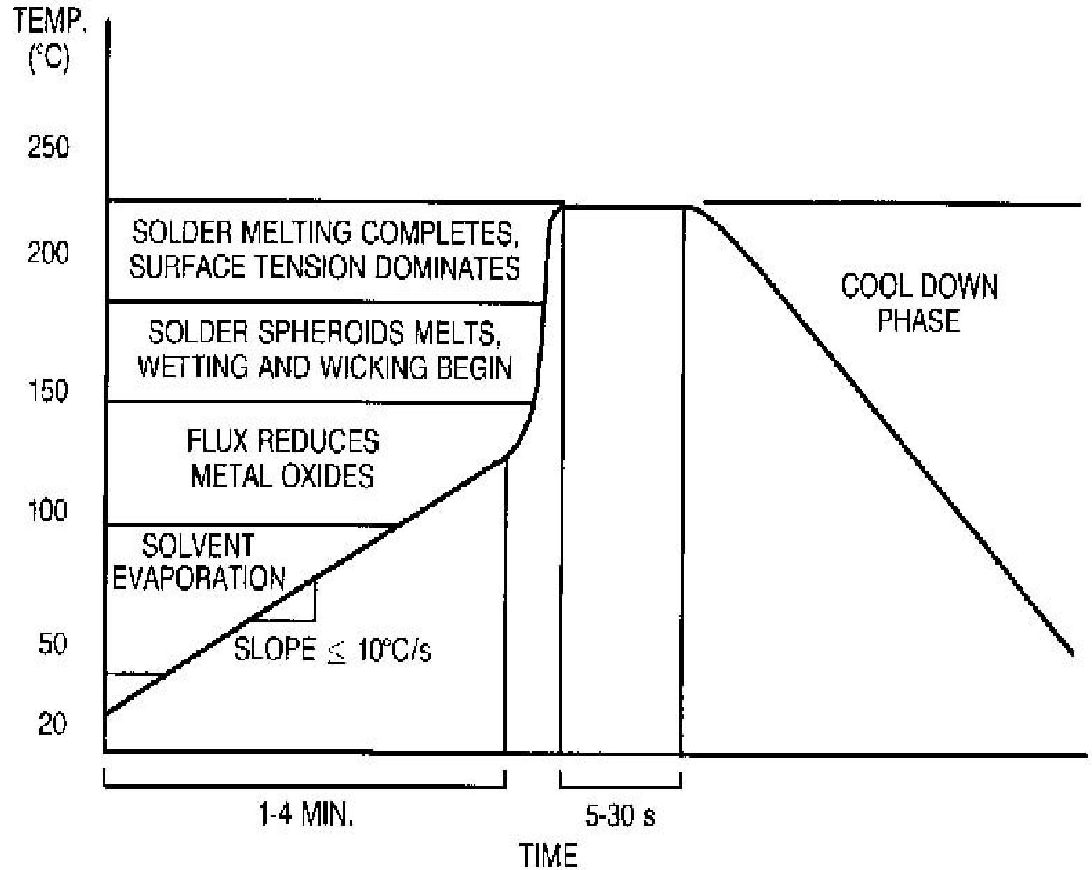
Paste and printing

- Solder paste has tiny metal spheres of the alloy mixed with flux, solvents, and thixotropic materials
- Methods of applying solder paste:
  - Stencil printing
  - Syringe dispensing
- Most influential step affecting yield



Reflow

- Once parts have been placed on the solder paste bricks, the entire board is placed in an oven and taken through a temperature profile
  - Linear oven
  - Multi-stage oven



Other steps after reflow

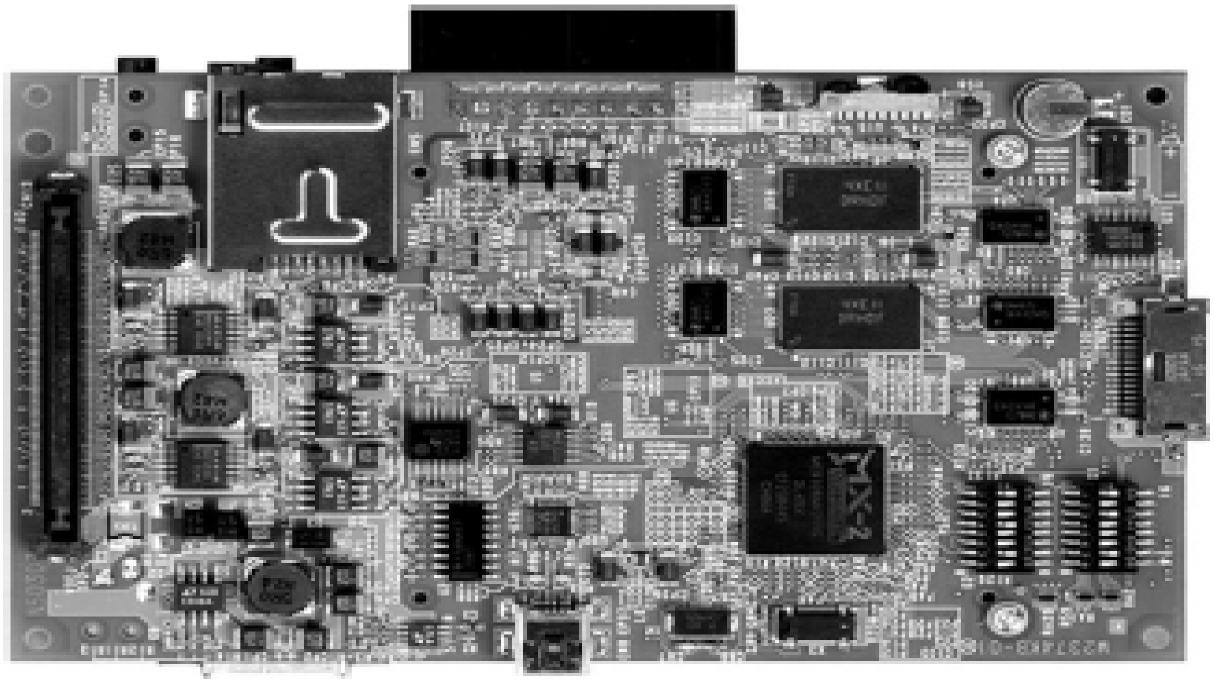
- Inspection/Test
  - Look for wrong/misplaced components and poor solder joints
- Rework/Backload
  - Fix problems and add parts that can't survive the high temperature of the reflow oven
- Cleaning
  - Wash to remove flux residues

Do it by yourself (large components/large pitch)

- Dispense (usually SnPb solder paste)
  - Use a robust paste with a wide process window
    - Alpha WS609(if you can clean the board or don’t care about long term reliability)
    - Kester R244 if you can’t clean
- Hand place components with tweezers
  - don’t let paste dry out
  - don’t push down too hard
  - always use ESD protection
- Hot plate (or hot air solder system)
  - only needs to be molten (~200C) for 60-90s
- Clean, if necessary

Printed Circuit Board

- Result after the mounting



Advantages of SMT

- smaller parts
- denser layout
- cheaper pcbs (no holes to drill)
- improved shock and vibration characteristics
- improved frequency response
- easier to shield from EMI / RFI
- easier to automate manufacturing

Disadvantages of SMT

- more heat generated
- small clearance makes cleaning difficult
- visual inspection difficult
- good joint formation important for mechanical reliability of assembly
- harder to hand assemble
- greater number of different materials to match CTE’s

Printed Circuit Board

- PCB design is affected by the typical problems of the design of complex and heterogeneous systems
  - A board can integrate a processor, with some software to be developed
  - It is important to organized the groups in order to work in parallel till the final phases of integration and system verification
  - An important step is the partitioning of the functionalities not only in terms of hw and sw, but also w.r.t. respect to the single devices
    - Precise identification of the input and output signals of each components and mapping of the logical signals onto the physical pins of the chip (*pinout*)

Printed Circuit Board

- PCB design criteria
  - The partitioning is not only driven by the functionalities
    - Number of signals necessary to connect two different chips
      - The number of pins leads to complex boards, rising the cost
    - Bandwidth of the signals
      - When operating at high frequency it is better (if possible) to keep the sections that need to communicate on a single chip
    - Delays of signals
      - Length and width of wires are characterized by time constants much higher than those of signals flowing internally to the chip
  - After identifying the system partitions and the pin-out, the designers and the board developers can work in parallel

Printed Circuit Board

- The problem of testing
  - *Boundary scan testing*
    - Methodology that uses a scan chain of cells (Flip-Flops) and multiplexers, that encompasses all the pins of all (or part of) the devices mounted on a board
      - *Normal operation*. The registers of the scan chain are connected to the actual I/O of the systems
      - *Test mode.* The registers are connected to realize a shift register (scan chain), that allows to carry the desired logic vale on a specific pin or, dually, to read a logic value and to expose it externally

      - The same approach is used at the level of logic blocks, for the testing of the SoC

Printed Circuit Board

- Boundary scan testing at board level