

SATURN SOLUTIONS

ELECTRONIC DESIGN AND MANUFACTURE

The Design Method

Initial Steps

In any product design, the first steps are to explore a market for your product and a need that it will fulfil in that market. You should then identify existing (competing) products and find a position alongside these for your new product.

Specification

Having explored the market place you will need to create a specification for your product. This will essentially contain your instructions to the designer outlining the elements that the new product should possess. A specification starts out as a list of ideas and requirements that describe a device or project. This gets trimmed down to something more exact and explicit over time. It will help the designer if you give your reasons for wanting the product to perform certain tasks and how **you** would expect the tasks to be performed. You should also list which elements are 'required' and which are 'desired' as it will sometimes be necessary (for any number of reasons / constraints) to make changes during the design phase. These lists, which should be kept separate from the main specification, will help the designer to have a clearer understanding of your requirements.

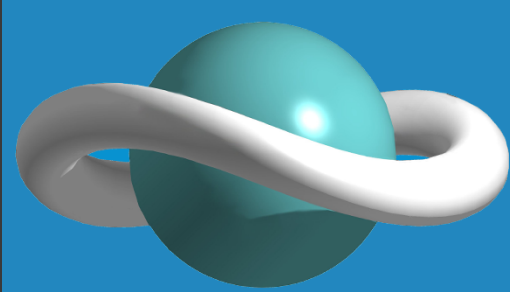
So the specification goes through two phases. In the first phase it describes the product as desired and, in the second phase it describes the product as required. The product specification, with your designers input, describes the product as it will be produced.

Circuit Design

The Circuit Diagram or Schematic is a 'map' of the components and inter-connections in its most easily readable form. The diagram is constructed using one of many standard CAD packages available and it must show exactly the construction of the printed circuit that is designed from it. Therefore, the schematic CAD and printed circuit board CAD are linked through a Net List and changes to one must always be carried out on the other. The circuit diagram will reference each component on the PCB and will list pin numbers for each connection. A good circuit diagram will include as much information as possible to help the reader understand the circuit.

The designer will need to take various things into consideration when producing the circuit including: -

1. Component specifications.
2. Compatibility of components. (There may be interaction issues.)



SATURN SOLUTIONS

ELECTRONIC DESIGN AND MANUFACTURE

3. Orientation of components to best suit pin outs.
4. Type of package being used i.e. Surface Mount or Conventional. (Some SM components can have slightly different characteristics to their conventional equivalents.)
5. Availability of components.

Printed Circuit Board Layout

The Printed Circuit Board (PCB) is the body of material to which components are soldered. Components are connected via one or more layers of metal tracks etched onto and passing through the board. The details for the PCB are taken directly from the circuit diagram and are positioned and placed by the designer for the best (and hopefully simplest) result. This layout represents how the circuit will actually look when produced and, although the connections on the PCB will be identical to the circuit diagram, the two may look completely different. This is because the circuit diagram is intended to be 'readable' whilst the PCB layout is intended to be 'functional'. *There may be a lot of unfamiliar terms that you come across when talking about PCBs, so we've put them in a [glossary of terms](#) later on in this guide.*

The finalised CAD layout file (or Gerber file) is sent to a specialist PCB manufacturer for a small batch to be produced and returned a few days later for a prototype assembly run.

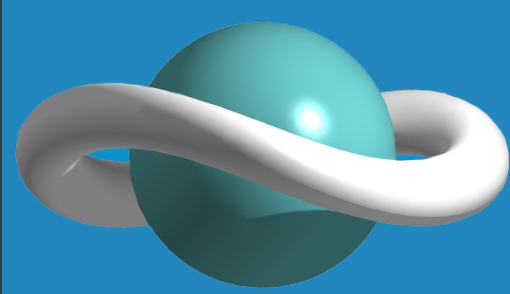
Prototypes

Two or three prototype circuit boards will now be produced by an engineer. During this time the circuitry is usually tested in stages following a complete test procedure as the PCB is assembled.

Test procedures for a prototype board

The perfect partner to a good specification is a good test procedure. There are three types of test procedure to discuss at this point.

- i. Debug Test procedure (developed by the designer) - This procedure will focus on probable trouble spots in a design. This will isolate any faults so that they can be rectified individually. The debug test will also include a complete functionality test with reference to the customer specification.



SATURN SOLUTIONS

ELECTRONIC DESIGN AND MANUFACTURE

ii. Incoming Test Procedure (developed by the customer) - This will again test functionality with reference to the customer specification. The aim of this test is to prove that the design meets the specification exactly.

iii. Manufacturing Test procedure (developed by the designer / customer / manufacturer) - This procedure is heavily dependent on the manufacturer's production techniques and test equipment. It is driven by economic motives and is aimed at reducing manufacturing costs by reducing failure and returns rates.

Pilot run

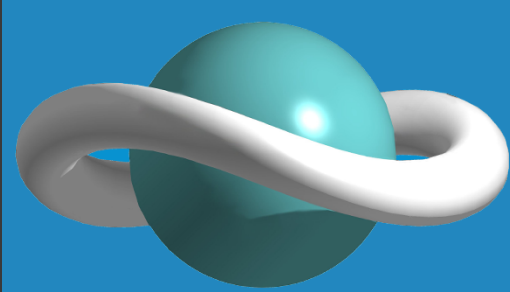
As an additional test a pilot run is sometimes required where a small number of units (say 20) are tested in the field. This is called a beta test and it is also a chance to consider how the design will lend itself to full scale manufacture and to check whether associated documentation is useable.

Production

The pilot run may reveal the need for changes to the design before it can be described as a finished product. Some designers (like Saturn) handle the full-scale production of products for their customers. Some customers prefer to do their own manufacturing or to use another 'preferred' sub-contractor.

What to do now?

If you have a requirement for a product design and / or manufacture, then email us or [contact us](#) through the enquiry form and we'll arrange to come and see you. If you would like more information, please call and put your questions directly to us. We will endeavour to give you sound, confidential, free advice with no obligation to proceed any further.



SATURN SOLUTIONS

ELECTRONIC DESIGN AND MANUFACTURE

Glossary of Terms

-

General

1. **CAD** - Computer Aided Design.
2. **CAM** - Computer Aided Manufacture.
3. **Pin Out** - The arrangement of inputs and outputs on an Integrated Circuit.
4. **Gerber File** - Gerber is a format which store all the relative coordinates of a PCB for production by the PCB manufacturer. It has been adopted as the industry standard because of its flexibility and reliability.

PCB Related Terms

1. **Conventional** - A rigid PCB with wire-leaded components mounted on only one side of the board with all the leads inserted through holes and soldered into position. Conventional assembly is generally easier to analyse and repair than surface mount.
2. **Surface Mount Technology (SMT)** - A PCB with tag leaded components soldered flush to solder pads. Holes are still required but not for component leads. Surface mount is generally more compact than conventional and is more suited to automated assembly.
3. **Mixed** - A mixture of surface mount and conventional components. There are disadvantages associated with the different soldering techniques required on this type of assembly.
4. **Hybrid** - A PCB that has both tracks and some components (normally resistors) already printed on it, ready to receive either or both surface mount and/or conventional devices.
5. **Double sided** - A PCB laminate with tracks on both sides, normally with plated through holes connecting circuitry on the two sides together.
6. **Plated Through (Thru) Hole (PTH)** - The holes are copper plated providing a circuit between sides of the PCB.
7. **Double sided Assembly** - Mounting components on both sides of a PCB. This would normally only be appropriate for SMT or mixed component assemblies.
8. **Multi-layer** - A PCB may be manufactured with more than two layers of copper tracks by sandwiching tracks within the laminate. This enables much more complex circuitry to be incorporated into a PCB but the cost reflects the number of layers.
9. **Gold plated** - Some parts of a PCB may be gold plated for use as contact pads. (Usually pads at the edge of the board.)
10. **Flexible PCB** - A flexible circuit widely used with membrane keyboards and circuits required to fit into small and awkward spaces.
11. **Chip On Board (COB)** - An IC is attached directly to a PCB with bond out wires connected directly to the surface of the board. (The IC has no casing or legs connected.) The chip is then protected with a layer of black epoxy. This type of technology is used mainly for cost saving in mass production applications.
12. **Daughter board** - A PCB mounted (piggy backed) onto another circuit board.