The Schematic Capture tool is part of a suite of Computer Aided Design (CAD) tools which assist engineers in the design of modern circuits. The CAD tool can be used to “capture” a virtual representation of a digital circuit on a computer, which can then be analyzed and simulated before an actual circuit board is created. There are many varieties of Schematic Capture CAD tools, allowing designers to create anything from complex VLSI circuitry which can be simulated to fully, to Microprocessor based boards, which cannot!

The Goal
The Block Diagram

- Every schematic begins with a block diagram
- The Block Diagram gives an architectural overview and shows the data paths and interrelationships between components of a system

Example of a Schematic

Example of a Schematic
Requirements

- There are many Schematic Capture CAD tools available to designers, providing:
  - Good user-interface, allowing the designer to seamlessly connect and modify different components
  - Good library support, which not only allows for creation of complex libraries, but also their usage in a hierarchical fashion
  - Electrical Rule Check
    - No two outputs are connected together
    - Inputs aren’t left floating
    - Fan-in and Fan-out limitations
  - Netlist generation and management
    - Enabling import/export to other modules such as simulators, or other CAD tools

Simulation

- Schematic Capture tools also allow for simulation of the electrical components
- However, with a microprocessor based design, there is no way of simulating the behavior of the circuit, in particular that of the microprocessor
- Instead, we design as carefully as possible, and hope for the best!
- Most designs go through at least one phase of redesign. This is why we first make a prototype of the system and then address all the changes in the production run of the board

Creating Libraries

- Many CAD packages come equipped with a wide array of component libraries, which include everything from resistors and capacitors, to integrated circuits
- Chances are, however, that you will need to create a library for most of your ICs, including the microprocessor from scratch
- Creating libraries is time consuming, but of crucial importance, since it is the building blocks of your schematic
- Components in a library must be checked and re-checked to ensure pins aren’t mislabeled and that there are no other errors
A component in a library is simply a symbolic representation of an integrated circuit. The picture to the right shows the 6809GT8 microcontroller’s pinouts. This image can be represented in one of two ways as a component in a schematic library:

- **Pin-to-pin Representation**
- **Grouped by Functionality**
Pin Properties

- Each pin of the component has a set of parameters that needs to be set at the time of creating the library component.
- This is important later on when we run the Electrical Rule Check (ERC) to ensure that no two outputs are connected to each other, and Power and Ground pins aren’t shorted.

Naming conventions

- Pin polarities, and their I/O characteristics are outlined in the processor spec. For example:
  - The reset signal RST# is active low, and is designated as such on the schematic with the '#' sign.
  - TXD is an output, although this property is not visible on the schematic.

Starting your schematic design
Adding components

Annotating the Schematic

- Annotating the schematic assigns a unique number to every component which can then be used to create a Bill Of Materials (BOM) which is then further used for manufacturing and component placement.

**RP Resistor Pack**
- R Resistor
- C Capacitor
- L Inductor and ferrite beads
- D Diode
- Q Transistor
- Y Crystal
- OSC Oscillator
- H Header
- J Connector
- RL Relay
- U Other components

Annotated Schematic
Electrical Rule Check

- The ERC is performed to ensure that a design meets electrical rule guidelines and that a valid netlist can be generated.
- The ERC checks both for electrical errors such as two output pins being connected together, and for drafting errors such as duplicate designators.
- Once an ERC has completed successfully, a netlist is generated.
- A Netlist is a text file listing all the connections between components. It is used to transfer the design between different CAD tools, and also used to import the design into a PCB document.

Bill of Materials

- The Bill of Materials (BOM) is an ordered list of part numbers along with their designators.
- It is the complete list of all the items that go into making the finished product, including the PCB board itself, casing, etc.
- It includes but is not limited to the manufacturer’s part number, quantity, description, and assembly notes for each component.