CP323505

Fusion 360 and EAGLE – Mechanical Engineering Meets the World of Electronics

Richard Hammerl
Autodesk GmbH

Learning Objectives

- Gain insight into the fundamentals of electronic design
- Learn about the fundamentals of Printed Circuits Boards design
- Learn about the manufacturing process of PCBs
- Discover how Fusion and EAGLE unifies MCAD and ECAD

Description

Many of the products we design and finally use are developed in mechanical CAD software. However, in almost all products, you will also find an electronic part. Without, most products would be pointless. With the two Autodesk products Fusion 360 software and EAGLE software, these two worlds grow together. This class will provide insight into the world of electronics. We intend to explain very basic things: What is a printed circuit board (PCB) and how is it manufactured? Which electronic components are used? How is an electronic layout created in EAGLE and how can a mechanical engineer use, change, and create a layout in Fusion 360? These basic questions will be explained using some illustrative examples.

Speaker

Richard Hammerl a graduated mechanical engineer came across EAGLE in 1994. It was the year he started to work for CadSoft Computer GmbH, the inventor and distributor of EAGLE, the well-known ECAD software tool. It was version 2.6 for DOS when he learned EAGLE by reading the manual and taking customer support calls from day one on. Meanwhile this ECAD system is under the wings of Autodesk and experienced a lot of improvements. Still with EAGLE, Richard Hammerl is a technical specialist for electronics and has a deep knowledge of electronics design software.

Email: richard.hammerl@autodesk.com

Table of Contents

History of MCAD and ECAD

Mechanical Design

If one looks back to the middle of the last century and follows the development in mechanical design and also in electronics development, one can find parallels. For a long time the engineering drawings were drawn by hand on paper. Calculations were made separately and the construction was designed accordingly.
Later there was the possibility to work computer-aided. In the beginning the computer was only used for drawing, later the programs became more and more versatile. Away from the pure drawing to the support of 3D representation, simulation of forces, dynamic loads, thermal analyses and much more.

Electronic Design

In the electronics sector we see a similar path. The idea of an electronic project drawn as a schematic on paper, transferred this project into a real design. Printed Circuits Boards were much more expensive than they are now, a lot of devices were built on the fly and manually wired.
Now we see software for Electronic Design. Draw the schematic, check functionality, create the PCB, let the software help you in finding the ideal routing paths of your signals. Generate manufacturing data and have the PCB produced. All done in one seamless workflow.
Further information is available via the following link:

About the History of PCBs.

→https://www.autodesk.com/products/eagle/blog/history-of-pcbs/

Electronics and Mechanics – Two Different Worlds?

Historically I would say yes, but meanwhile most of our products include electronics. Most of our products are getting smarter, smaller, more complex. Think of computers, smart phones, cars… So these two worlds belong together and in product development you have to collaborate. This is the reason why we made EAGLE and Fusion 360 work together.

Fundamentals of Electronics

Electronics Design, Drawing a Schematic, Layout Design, Trace Routing, Resistors, Transistors, Schematic Symbols, Footprints, Printed Circuits Boards – What is this all about? Let's take a look into the world of electronics.

Electronic Components

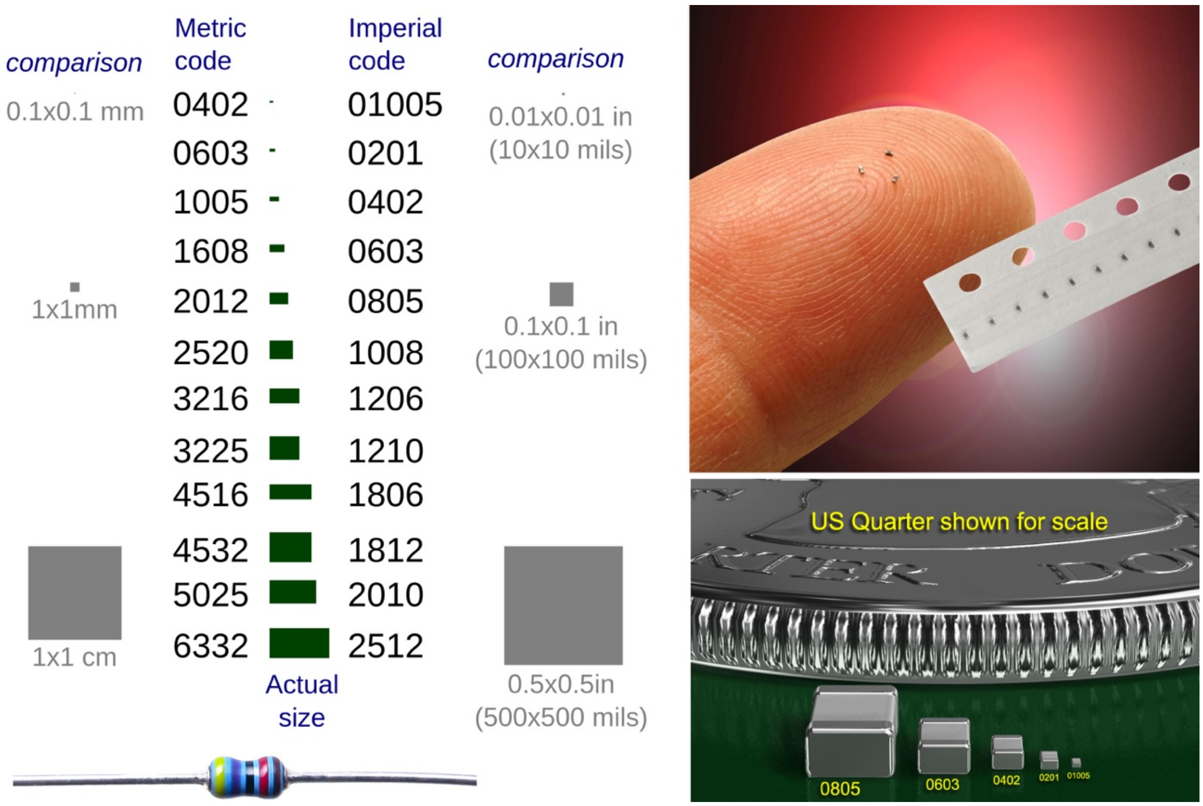
Resistors, Capacitors, Transistors, Inductors, LEDs, MOSFETs, Op-Amps, Diodes, Logic Gates and many more different components are available. As an electronic engineer you know all these expressions and what they describe. If you do not know, take a look into the web. There you will find all lot of information about them.
Further information is available in the Autodesk EAGLE blog via the following links:

How to Choose the Right Resistor?
→https://www.autodesk.com/products/eagle/blog/how-choose-right-resistor/
All about Capacitors
→https://www.autodesk.com/products/eagle/blog/everything-need-know-capacitors/
What Is an Inductor, In Plain English?
→https://www.autodesk.com/products/eagle/blog/inductor-plain-english/
How does a diode and LED work?
→https://www.autodesk.com/products/eagle/blog/diode-led-work/
What is a Transistor?
→https://www.autodesk.com/products/eagle/blog/transistors-world-modern-electrons/
How Transistors Changed Electronics Forever
→https://www.autodesk.com/products/eagle/blog/how-transistors-changed-electronics-forever/
Op-Amps – A beginners Guide
→https://www.autodesk.com/products/eagle/blog/op-amps-beginners-guide/
How the Integrated Circuit Works: Everything You Need to Know
→https://www.autodesk.com/products/eagle/blog/integrated-circuit-moores-law/
How do Microcontrollers work?
→https://www.autodesk.com/products/eagle/blog/how-microcontrollers-work/
How Logic Gates Work in Digital Electronics
→https://www.autodesk.com/products/eagle/blog/you-shall-not-pass-how-logic-gates-work-in-digital-electronics/

There is another amazing development in electronics. As you know all our electronic devices are getting smarter and smarter and at the same time smaller and smaller.

Miniaturization of Electronics
→https://www.autodesk.com/products/eagle/blog/the-miniaturization-of-electronics/

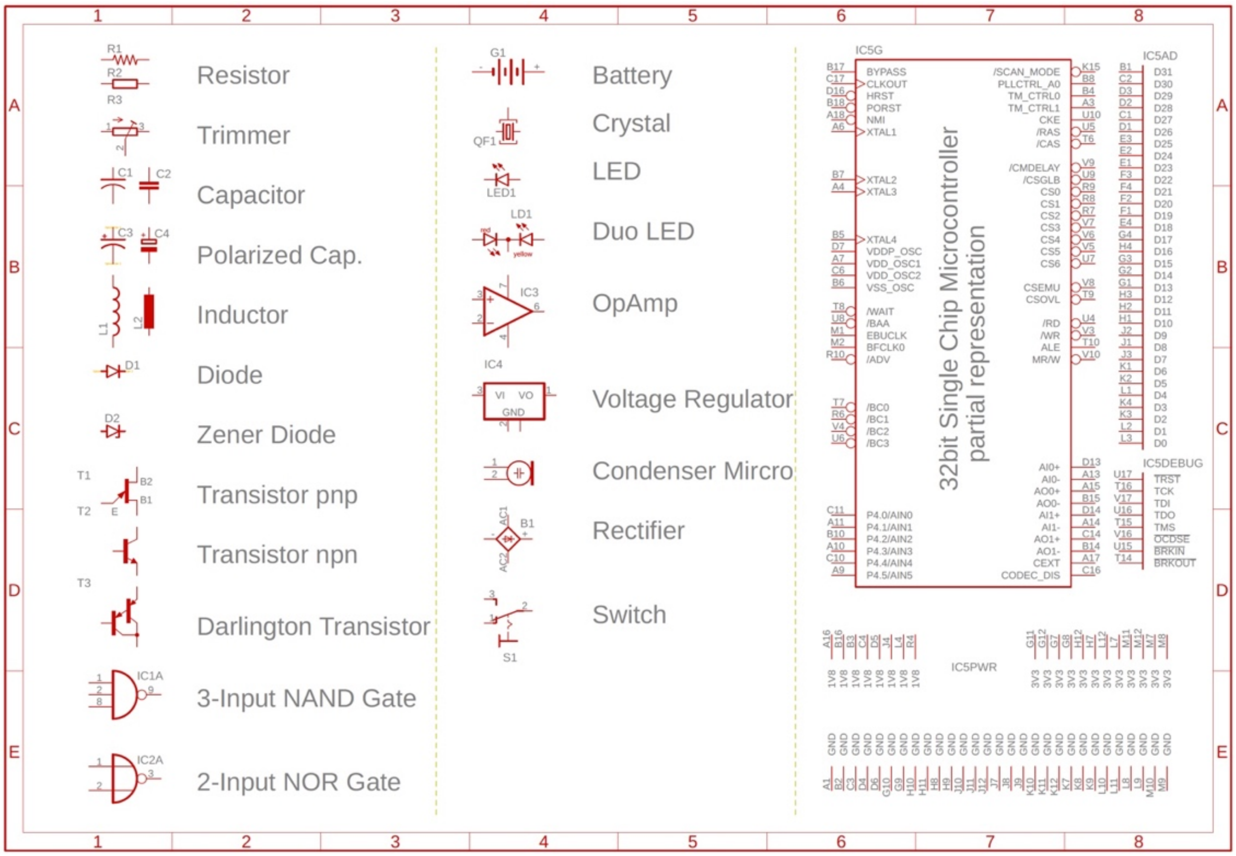
The following image shows an impressive change of size for electronic devices.



RESISTORS IN DIFFERENT PACKAGE SIZES

Logical Representation of a Component in a Schematic Drawing

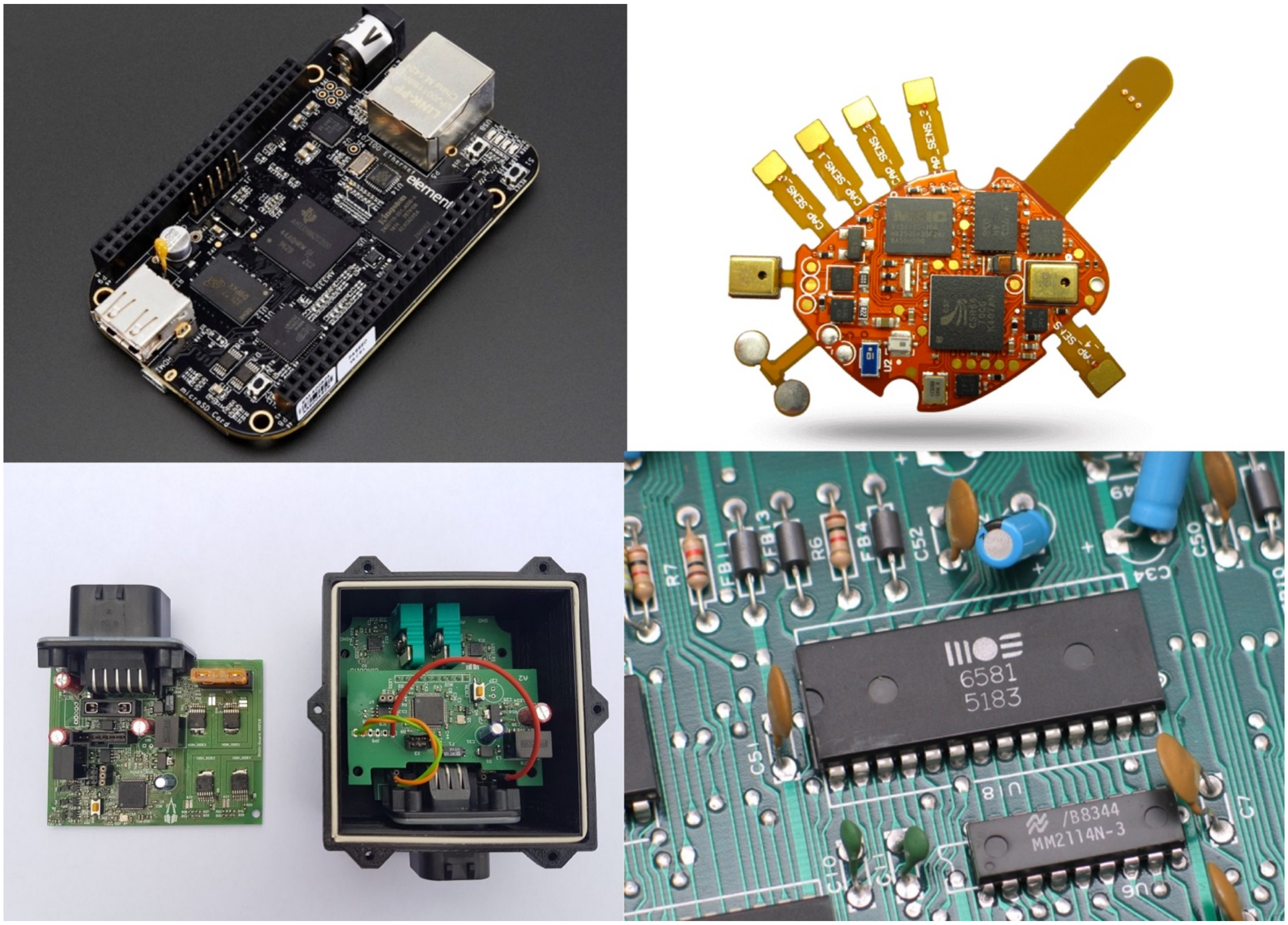
The Schematic is a drawing that contains symbols, the logical representation of components. Some examples:



SYMBOLS USED IN THE SCHEMATIC DRAWING

How Do Printed Circuits Boards Look Like?

PCBs are available in all kind of shapes. They can be rigid or flexible or a combination of both. Some boards have components on side or even on both sides (top and bottom side). The vast majority is two layers. This means copper traces on top and bottom. Depending on complexity the can also have more copper layers, so-called inner layers. EAGLE supports up to 16 signal layers.



PRINTED CIRCUIT BOARDS

Further information is available via the following link:

[Printed Circuit Boards](https://www.autodesk.com/products/eagle/blog/printed-circuit-boards-10000-feet-introduction-electronics-beginners/)

→ https://www.autodesk.com/products/eagle/blog/printed-circuit-boards-10000-feet-introduction-electronics-beginners/

How is a PCB manufactured?

The basic data format for industrial manufacturing of a Printed Circuits Board is Gerber data and Excellon data for drilling. Please follow these links to get further information:

Gerber and Drill data

→ https://www.autodesk.com/products/eagle/blog/gerber-nc-drill-pcb-manufacturing-basics-1/

PCB manufacturing.

→ https://www.autodesk.com/products/eagle/blog/pcb-manufacturing/

Alternative options for PCB manufacturing:

→ https://www.autodesk.com/products/eagle/blog/additive-pcb-manufacturing-desktop/

Making a PCB – PCB Manufacture Step-by-Step

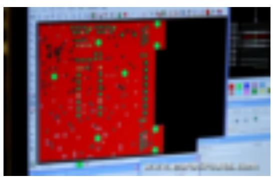
Thanks to Eurocircuits for sharing this on their web site. Click on the images for getting linked.

→ https://www.eurocircuits.com/making-a-pcb-pcb-manufacture-step-by-step/

Who are we and why have we made this film?

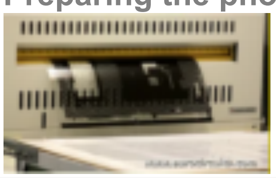
Eurocircuits are a European manufacturer of standard technology printed circuit boards (or PCBs). Our headquarters are in the picturesque Belgian town of Mechelen and our production units are near Aachen in Germany and at Eger in Hungary. We specialise in providing prototype and small batch PCBs for designers, product development departments, niche market electronics companies, universities and research establishments.

Front-end tool data preparation



The board designer has prepared his layout on a Computer Aided Design or CAD system. Each CAD system uses its own internal data format, so the PCB industry has developed a standard output format to transfer the layout data to the manufacturer. This is Extended Gerber or RS274X. The Gerber files define the copper tracking layers (4 in the job we are following) as well as the solder masks and component notations…

Preparing the photo tools

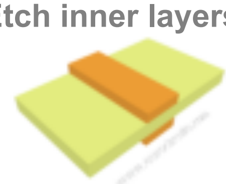


We use laser photoplotters in a temperature and humidity-controlled darkroom to make the films we will use later to image the PCBs. The photoplotter takes the board data and converts it into a pixel image. A laser writes this onto the film. The exposed film is automatically developed and unloaded for the operator.

Print inner layers

To produce the inner layers of our multilayer PCB, we start with a panel of laminate. Laminate is an epoxy resin and glass-fibre core with copper foil pre-bonded onto each side.

Etch inner layers



We remove the unwanted copper using a powerful alkaline solution to dissolve (or etch away) the exposed copper. The process is carefully controlled to ensure that the finished conductor widths are exactly as designed. But designers should be aware that thicker copper foils need wider spaces between the tracks. The operator checks carefully that all the unwanted copper has been etched away.

Register punch and Automatic Optical Inspection (AOI)

The inner core of our multilayer is now complete. Next we punch the registration holes we will use to align the inner layers to the outer layers. The operator loads the core into the optical punch which lines up the registration targets in the copper pattern and punches the registration holes.

Lay-up and bond

The outer layers of our multilayer consist of sheets of glass cloth pre-impregnated with uncured epoxy resin (prepreg) and a thin copper foil.

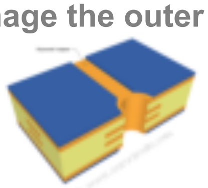
Drilling the PCB

Now we drill the holes for leaded components and the via holes that link the copper layers together. First we use an X-ray drill to locate targets in the copper of the inner layers. The machine drills registration holes to ensure that we will drill precisely through the centre of the inner layer pads.

Electroless copper deposition

The first step in the plating process is the chemical deposition of a very thin layer of copper on the hole walls.

Image the outer layers



We image the outer layers in a clean room to make sure that no dust gets onto the panel surface where it could cause a short or open circuit on the finished PCB.

Plating

Next we electroplate the boards with copper. The operator loads the panels onto the flight bars. He checks all the clamps to ensure a good electrical connection.

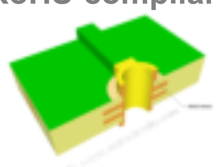
Etch outer layers

We have now plated the panel with 25 microns of copper through the hole and an additional 25 – 30 microns on the tracks and pads. The copper is covered with a thin layer of tin as an etch resist. Now we will remove the unwanted copper foil from the surface.

Apply solder mask

Most boards have an epoxy-ink solder mask printed onto each side to protect the copper surface and prevent solder shorting between components during assembly.

RoHS-compliant surface finishes – electroless gold over nickel



The copper component pads and holes have been left clear of solder mask. Now we apply a solderable surface finish to protect the copper until the components are soldered onto the board.

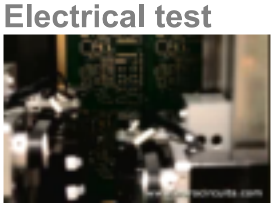
Plated gold edge connectors

For edge-connectors we electroplate hard gold. First the operator puts protective tape on the board above the connectors. Then he mounts the panel on a horizontal electroplating bath.

Silk-screen and cure

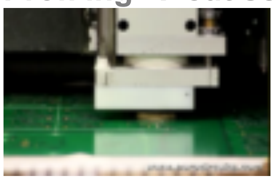
Most PCBs have a component legend to show which component goes where. Today we use ink-jet printers to image the legends direct from the board digital data.

Electrical test



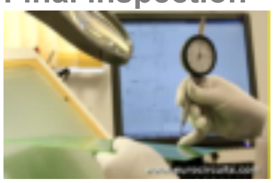
We electrically test every multilayer PCB against the original board data. Using a flying probe tester we check each net to ensure that it is complete (no open circuits) and does not short to any other net.

Profiling -V-cut scoring



The final manufacturing stage is to profile the PCBs and cut them out of the production panel.
Alternatively we can profile the individual circuits on a V-cut or scoring machine. This has two cutters, one above and one below the panel. Each cuts a v-shaped groove about 1/3 of the thickness of the material, leaving a thin web holding the PCBs together.

Final inspection



In the last step of the process a team of sharp-eyed inspectors give each PCB a final careful check-over.

What is EAGLE?

The name EAGLE is an acronym for **E**asily **A**pplicable **G**raphical **L**ayout **E**ditor. This also works in German language: **E**infach **A**nzuwendender **G**rafischer **L**ayout **E**ditor.

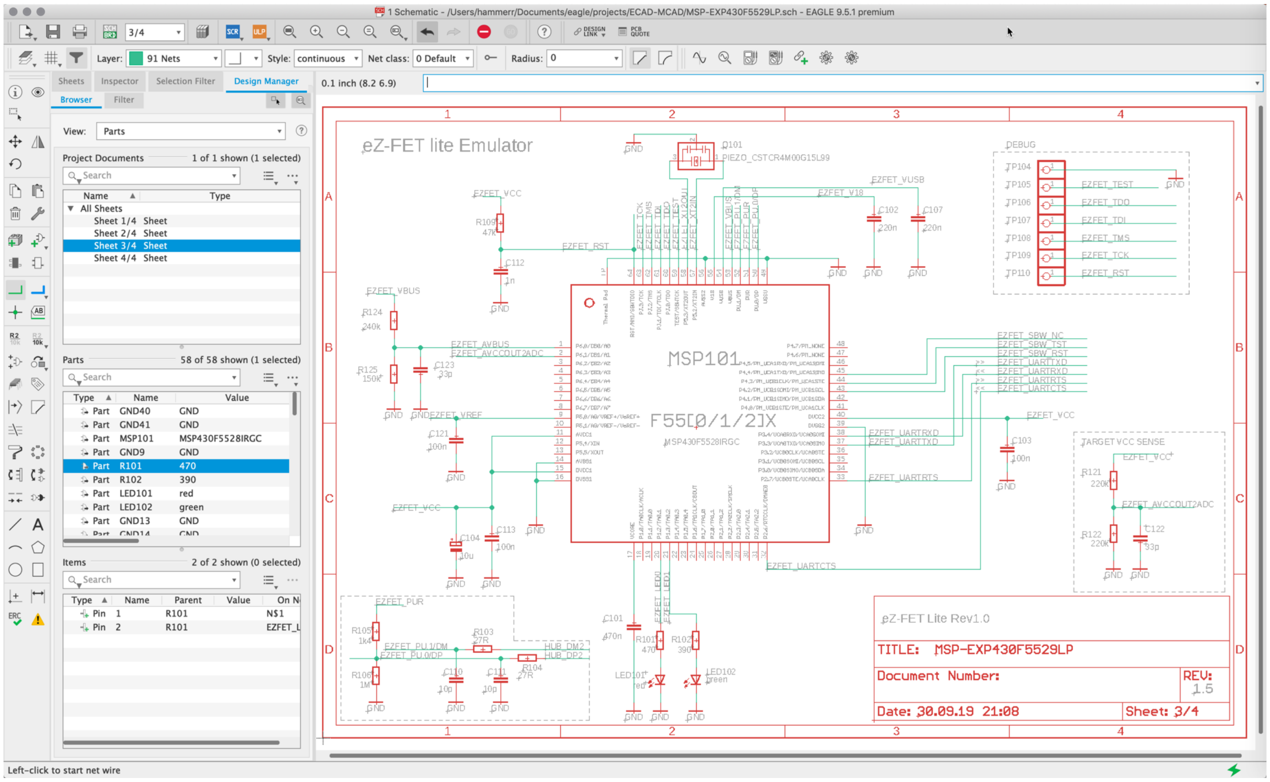
The software was invented in the late 80s in Germany. It is used for electronics development in all kind of industries and in almost all areas of electronics. EAGLE consists of several work spaces I want to introduce in the following sections.

For general information please follow this link:

Circuit Design Software
→ https://www.autodesk.com/solutions/circuit-design-software

Schematic Workspace

The Schematic is used to bring your idea of an electronic device onto "paper". In this case of course it is not real paper, but you draw it in the Schematic Workspace, possibly on several Schematic sheets depending on its complexity.



The EAGLE Schematic Workspace

How is this done? Further information is available via the following links:

[Designing your schematic like a pro](https://www.autodesk.com/products/eagle/blog/top-10-tips-draw-next-schematic-design-like-pro/)
→ https://www.autodesk.com/products/eagle/blog/top-10-tips-draw-next-schematic-design-like-pro/

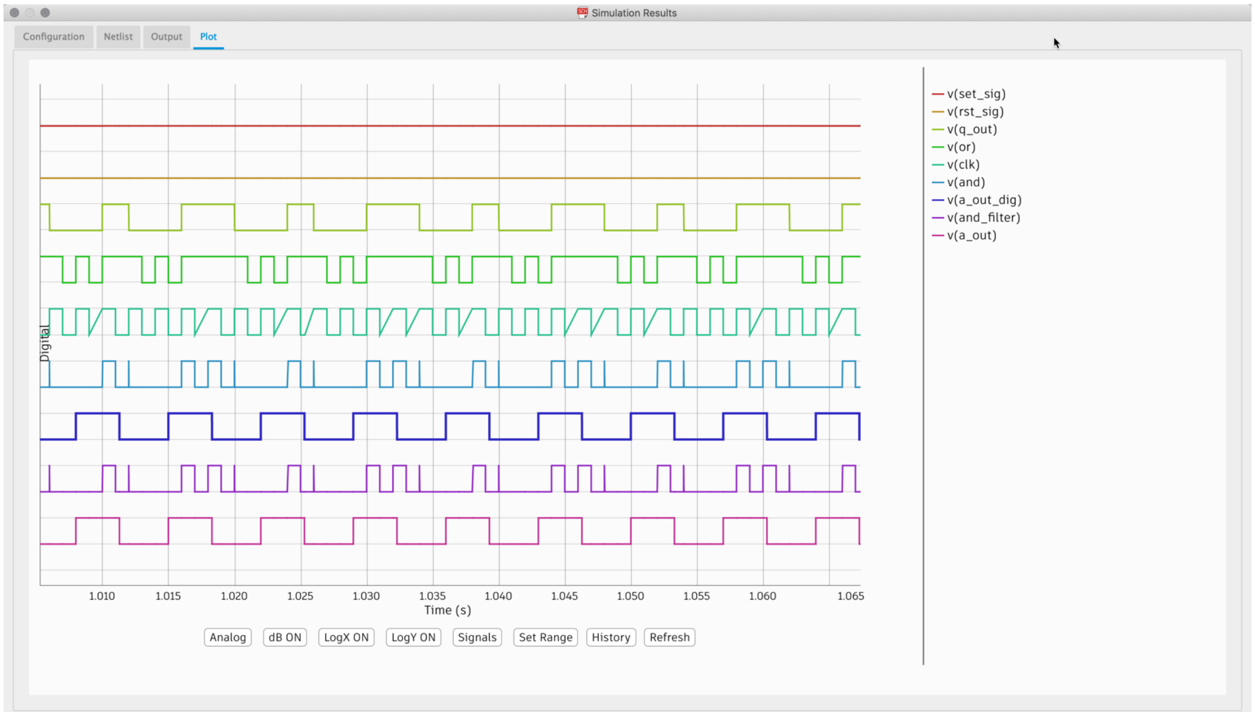
[Search, Place, and Move](https://www.autodesk.com/products/eagle/blog/schematic-basics-part-1/)
→ https://www.autodesk.com/products/eagle/blog/schematic-basics-part-1/

[Nets and Values](https://www.autodesk.com/products/eagle/blog/schematic-basics-part-2-nets-and-values/)
→ https://www.autodesk.com/products/eagle/blog/schematic-basics-part-2-nets-and-values/

[Electrical Rule Check](https://www.autodesk.com/products/eagle/blog/schematic-basics-part-3-erc/)
→ https://www.autodesk.com/products/eagle/blog/schematic-basics-part-3-erc/

SPICE Simulation

Verification of the functionality of your schematic. Do the signals behave as expected? Are voltages and currents as expected? SPICE stands for **S**imulation **P**rogram with **I**ntegrated **C**ircuit **E**mphasis. The Schematic editor comes with an integrated SPICE simulator for analog, digital and mixed-mode simulation.



RESULTING PLOT OF SPICE SIMULATION

Please follow these links for a series of six blog posts for detailed explanation:

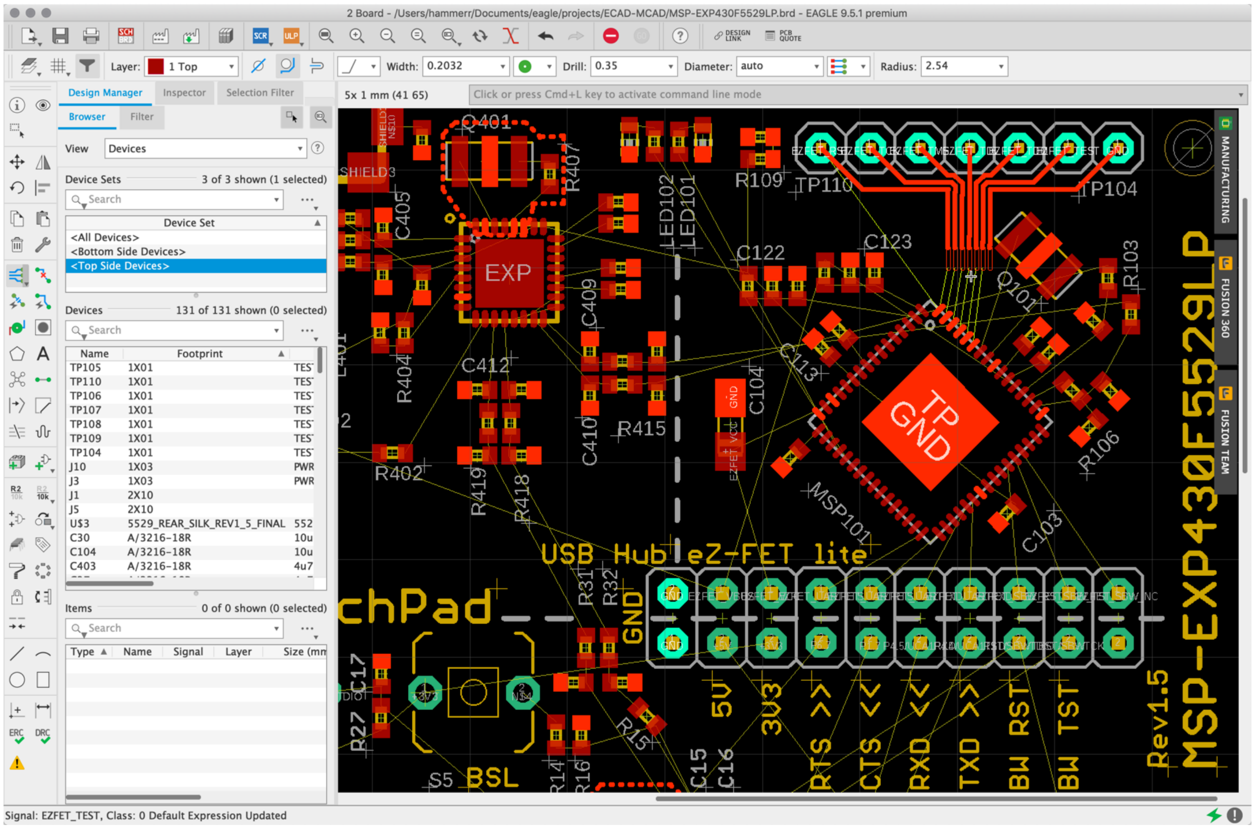
→ https://www.autodesk.com/products/eagle/blog/spice-simulation-part-1/
→ https://www.autodesk.com/products/eagle/blog/spice-simulation-part-2-operating-point-analysis/
→ https://www.autodesk.com/products/eagle/blog/spice-simulation-part-3-dc-ac-sweep-analysis/
→ https://www.autodesk.com/products/eagle/blog/spice-simulation-part-4-transient-analysis-spice-model-mapping/
→ https://www.autodesk.com/products/eagle/blog/spice-simulation-part-5-schematic-spice-mapping/
→ https://www.autodesk.com/products/eagle/blog/spice-simulation-part-6-make-eagle-libraries-spice-ready/

Layout Workspace

Once the schematic is done, the next step is layout design.
In the very beginning you may want to contact your board manufacturer and talk about design rules of the manufacturing process. Define the geometry of the PCB, decide about the layer stack up, place your components and route all signals in the board.

By the way: Define the board geometry: This is something you can do in EAGLE, or you can pull it from Fusion 360 into EAGLE. Here comes the connection to MCAD! You will find more information on this later in this document.

I want to mention one super convenient feature in EAGLE. It's the *Forward and Back Annotation*. It will keep your schematic and board in sync. Automatically and in real-time. Each change you do in the schematic will automatically be transferred into the layout and vice versa.



THE LAYOUT WORKSPACE: COMPONENTS PLACED, ROUTING TRACES JUST STARTED

Further information is available via the following links:

[Place your components](https://www.autodesk.com/products/eagle/blog/pcb-layout-basics-component-placement/)

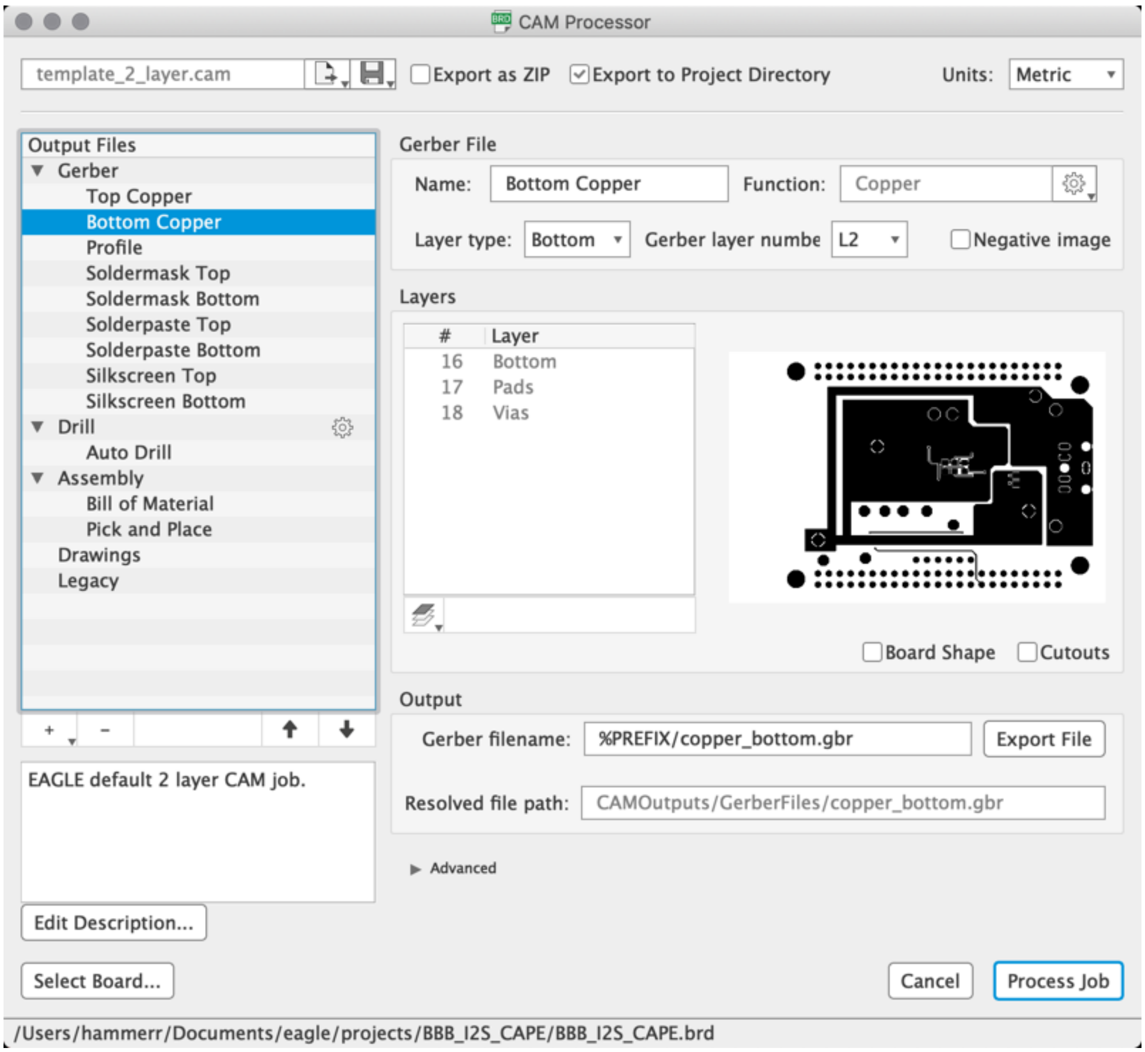
→ https://www.autodesk.com/products/eagle/blog/pcb-layout-basics-component-placement/

[Basics of Routing Traces](https://www.autodesk.com/products/eagle/blog/routing-autorouting-pcb-layout-basics-2/)

→ https://www.autodesk.com/products/eagle/blog/routing-autorouting-pcb-layout-basics-2/

CAM Processor

Next step in a typical electronic design process is to create manufacturing data for the PCB. As mentioned before, the CAM Processor creates Gerber data and Drill data for the board manufacturer. If you want to go for more than the bare circuit board, the CAM Processor will generate a bill of materials and Pick&Place data for an assembly service.



THE CAM PROCESSOR FOR GENERATING MANUFACTURING DATA

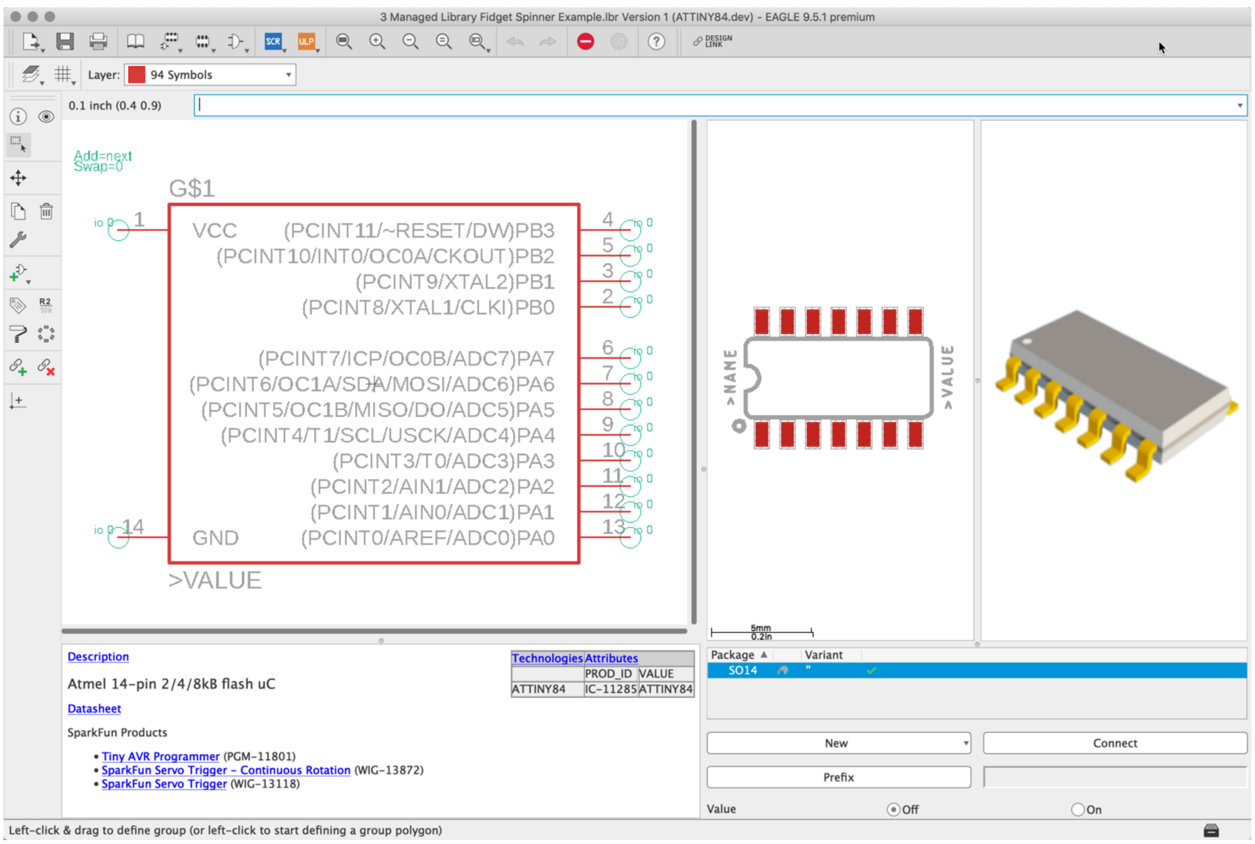
Further information is available via the following link:

[Basic questions before generating manufacturing data](https://www.autodesk.com/products/eagle/blog/5-questions-ask-throwing-money-first-pcb-manufacturer/)

→ https://www.autodesk.com/products/eagle/blog/5-questions-ask-throwing-money-first-pcb-manufacturer/

Library Workspace

The foundation for a flawless workflow from Schematic to Layout is anchored the libraries where all the components are defined. A symbol in the schematic is connected to a footprint used in the layout. The reference between pin in the schematic and pad/SMD in the layout is substantial for correct signal connections. Besides the internal pin/pad connection dimensions and sizes of the footprints must be accurate. For the connection into the mechanical world 3D models have to be assigned.



THE DEVICE EDITOR: SYMBOL ASSIGNED TO FOOTPRINT AND 3D PACKAGE

Further information is available via the following links:

Basic library work.

→https://www.autodesk.com/products/eagle/blog/library-basics-libraries-work-autodesk-eagle-make/

Creating your first package

→https://www.autodesk.com/products/eagle/blog/library-basics-part-1-creating-first-package-autodesk-eagle/

Creating your first symbol

→ https://www.autodesk.com/products/eagle/blog/library-basics-part-2-creating-first-symbol-autodesk-eagle/

Creating you first Device

→ https://www.autodesk.com/products/eagle/blog/library-basics-part-3-creating-first-device-autodesk-eagle/

Library management

→ https://www.autodesk.com/products/eagle/blog/library-management/

Online Library management
→ https://www.autodesk.com/products/eagle/blog/new-managed-online-libraries-use/
How to use Sparkfun and Adafruit libraries
→ https://www.autodesk.com/products/eagle/blog/library-basics-install-use-sparkfun-adafruit-libraries-autodesk-eagle/
Recent Updates in EAGLE Library Workspace
→ https://www.autodesk.com/products/eagle/blog/eagle-libraries-ui-ux/

ECAD meets MCAD

Collaboration

In the sections before you learned about basics in electronics. For a long time product development in electronics was not closely connected to mechanical development. At the end of the design process you could export data in DFX, IGES, or maybe STEP and sent this file to the mechanical guy. To be honest, in my opinion too inconvenient and so error-prone......

See these blogs about:

Failed ECAD-MCAD exchange file formats
→ https://www.autodesk.com/products/eagle/blog/complete-history-failed-ecad-mcad-exchange-file-formats/
Why ECAD-MCAD?
→ https://www.autodesk.com/products/eagle/blog/3-reasons-electronic-designers-need-mcad-collaboration/

EAGLE and Fusion 360 do a better job!
We do not convert data back and forth – we use a common data platform for this. Collaboration in both directions with immediate notification in case there are changes in the design.

Some more information about how this is done can be found in our Autodesk EAGLE channel. We are frequently updating the content with training videos or short tips and webinar recordings. All topics about workflow between EAGLE and Fusion 360 are collected in our ECAD/MCAD playlist:

Playlist ECAD to MCAD Unification
→ https://www.youtube.com/playlist?list=PL1rOC5j_Fyi6B0wRJw2GAVEjE_dNR7rlK

One of the videos shown in my AU class can be found here as well:

Defining a PCB Outline From an Enclosure in Fusion 360
→ https://www.youtube.com/watch?v=Ttld9rSPdVY