Fusion360 & EAGLE – Mechanical Engineering Meets the World of Electronics

Richard Hammerl

Technical Specialist Electronics

AUTODESK. UNIVERSITY

About the speaker

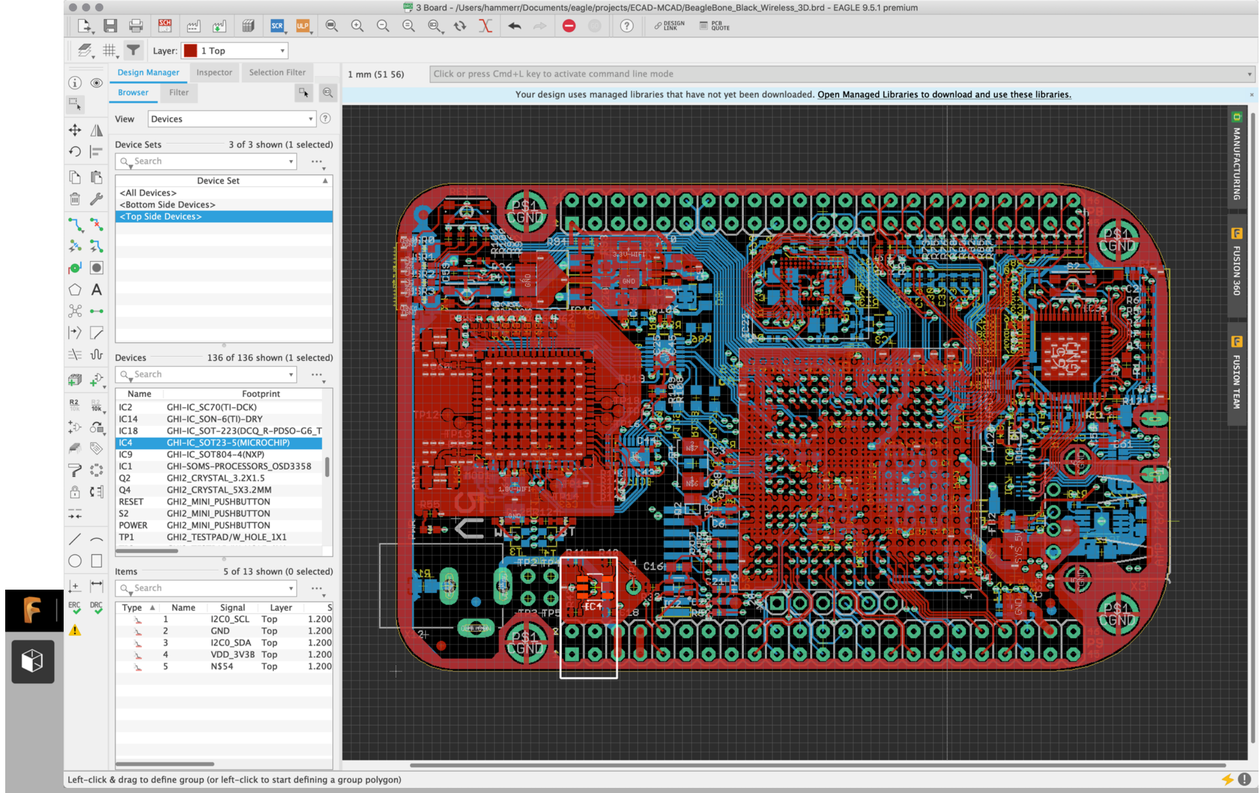
Richard Hammerl, Autodesk

EAGLE – since 1994 constant companion in the professional life of Richard Hammerl. He entered the world of electronics as a mechanical engineer and is familiar with all questions related to ECAD software.

After such a long time, the circle is now closing. Electronics development and mechanical design are moving closer together.

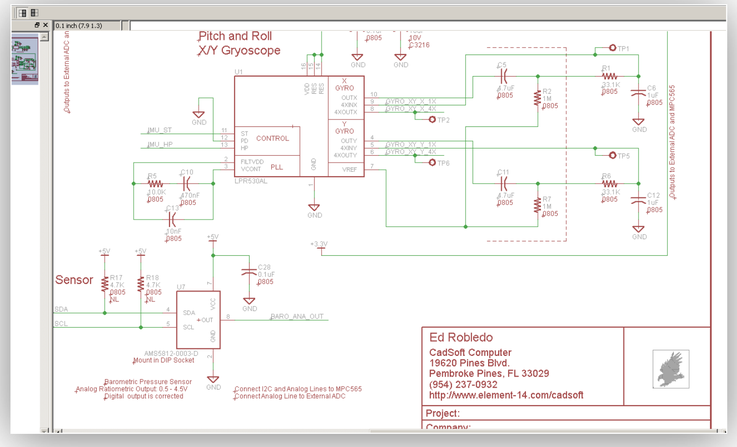
What is this class about?

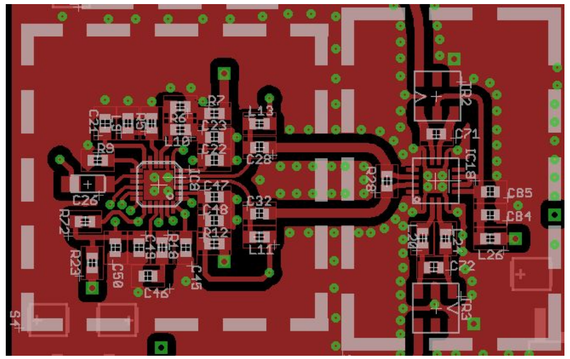
- Short history of EAGLE
- Electronics ?
- Fusion 360 and EAGLE
- What is EAGLE? What can it do?
- Cooperation Fusion and EAGLE
- Create printed circuit boards in Fusion
- Pull and Push with common database
- Changes in the Layout
- Data sharing with team members

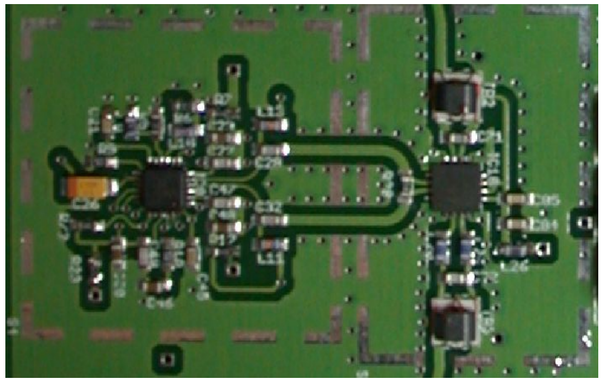


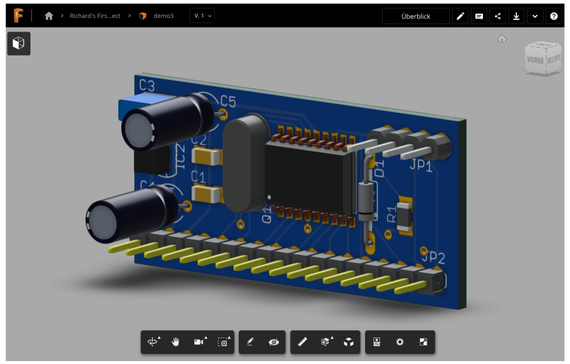
Intro

EAGLE







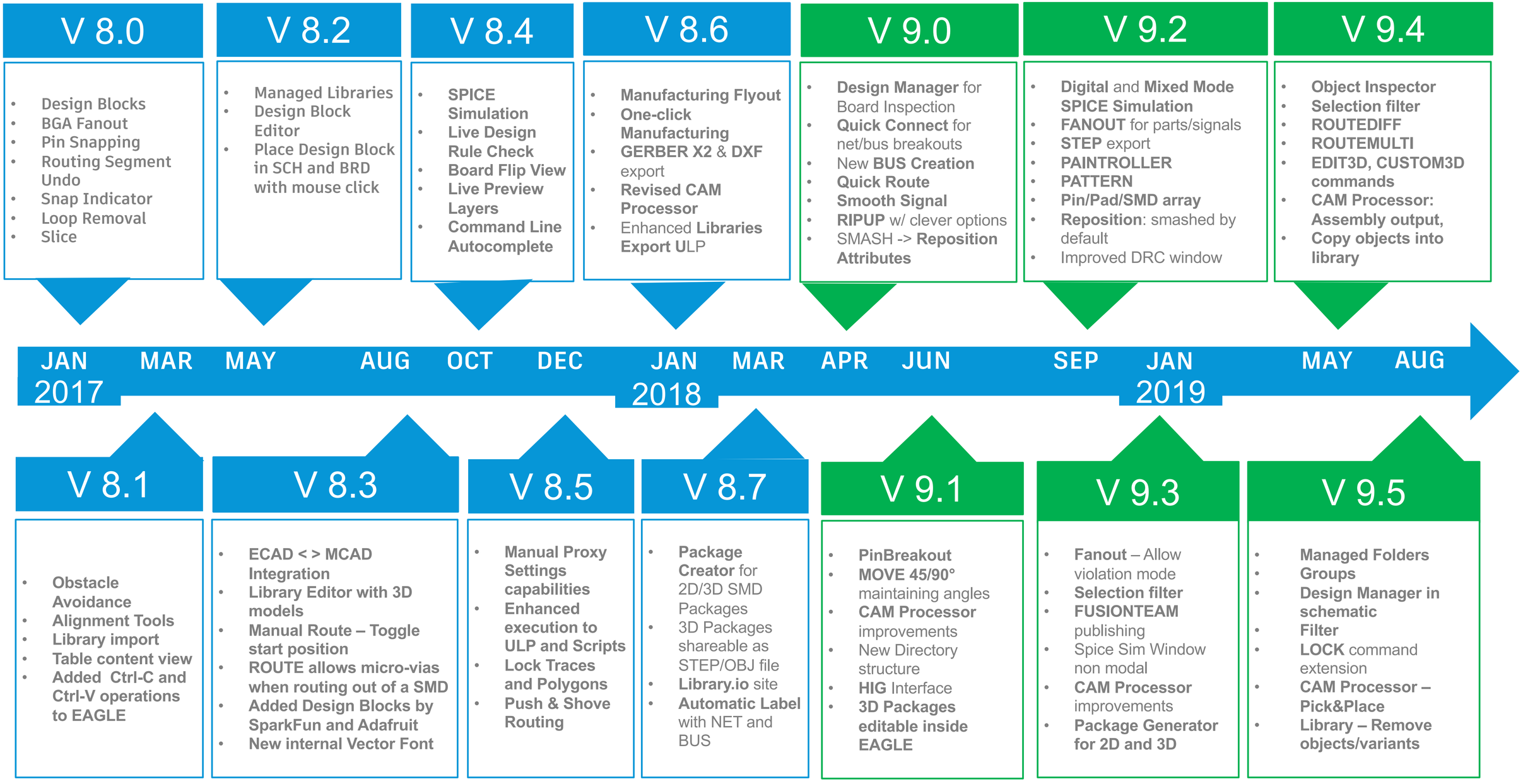


1988 - CadSoft Computer GmbH, Germany
1992 - Office in Fort Lauderdale, FL, USA
2009 - Acquired by Premier Farnell
2016 - Autodesk took EAGLE under its wings

SEIT
1988

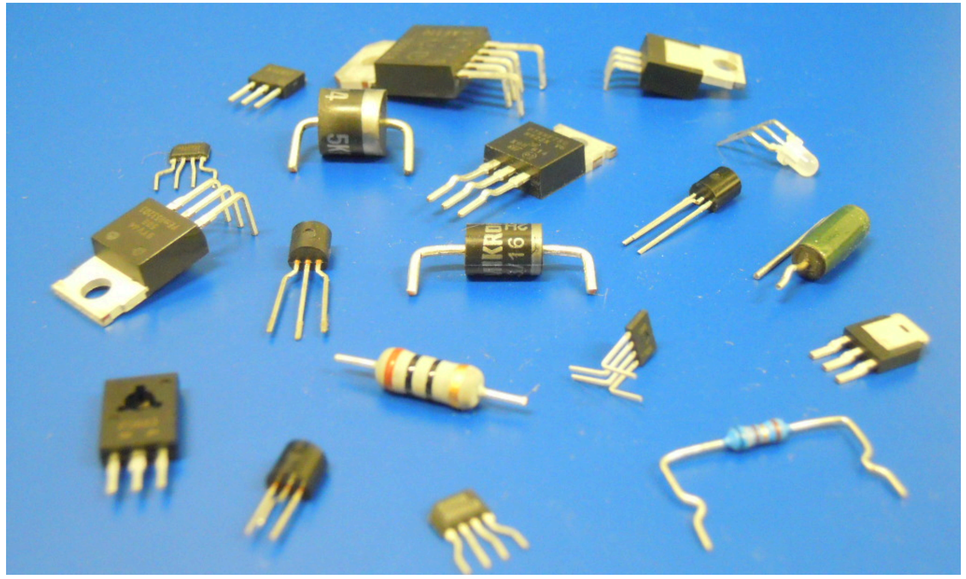
EASILY – **A**PPLICABLE – **G**RAPHICAL – **L**AYOUT - **E**DITOR

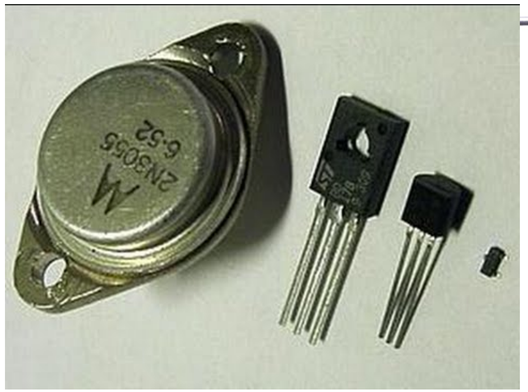
EINFACH – **A**NZUWENDENDER– **G**RAPHISCHER– **L**AYOUT - **E**DITOR

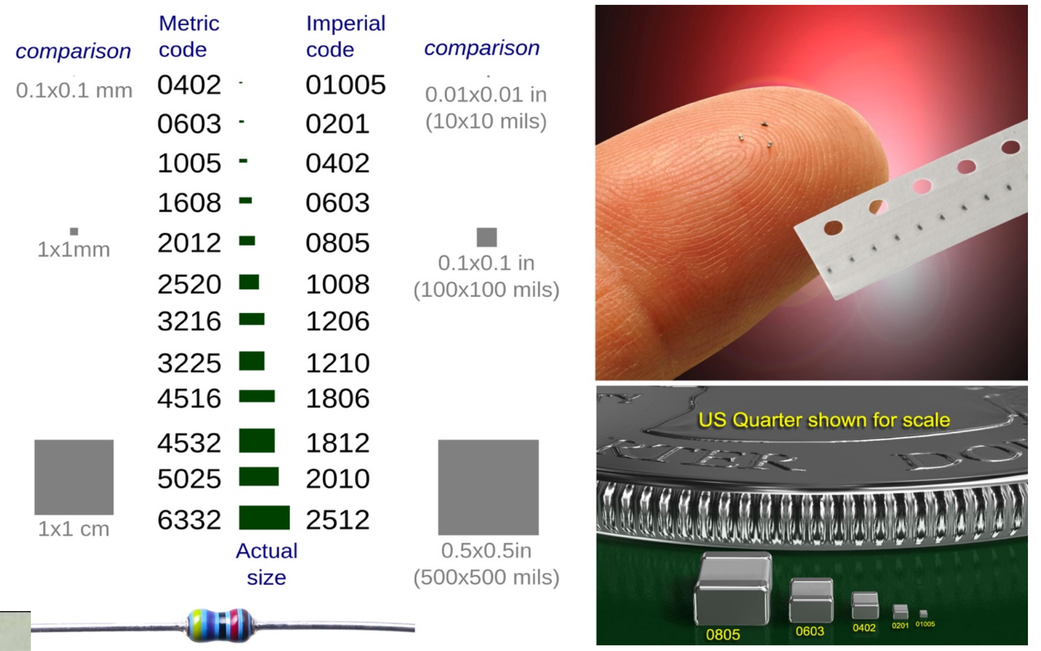

V 8.0
Design Blocks
BGA Fanout
Pin Snapping
Routing Segment Undo
Snap Indicator
Loop Removal
Slice
V 8.2
Managed Libraries
Design Block Editor
Place Design Block in SCH and BRD with mouse click
V 8.4
SPICE Simulation
Live Design Rule Check
Board Flip View
Live Preview Layers
Command Line Autocomplete
V 8.6
Manufacturing Flyout
One-click Manufacturing
GERBER X2 & DXF export
Revised CAM Processor
Enhanced Libraries Export ULP
V 9.0
Design Manager for Board Inspection
Quick Connect for net/bus breakouts
New BUS Creation
Quick Route
Smooth Signal
RIPUP w/ clever options
SMASH -> Reposition Attributes
V 9.2
Digital and Mixed Mode SPICE Simulation
FANOUT for parts/signals
STEP export
PAINTROLLER
PATTERN
Pin/Pad/SMD array
Reposition: smashed by default
Improved DRC window
V 9.4
Object Inspector
Selection filter
ROUTEDIFF
ROUTEMULTI
EDIT3D, CUSTOM3D commands
CAM Processor: Assembly output,
Copy objects into library
JAN
MAR
MAY
AUG
OCT
DEC
JAN
MAR
APR
JUN
SEP
JAN
MAY
AUG
2017
2018
2019
V 8.1
Obstacle Avoidance
Alignment Tools
Library import
Table content view
Added Ctrl-C and Ctrl-V operations to EAGLE
V 8.3
ECAD < > MCAD Integration
Library Editor with 3D models
Manual Route – Toggle start position
ROUTE allows micro-vias when routing out of a SMD
Added Design Blocks by SparkFun and Adafruit
New internal Vector Font
V 8.5
Manual Proxy Settings capabilities
Enhanced execution to ULP and Scripts
Lock Traces and Polygons
Push & Shove Routing
V 8.7
Package Creator for 2D/3D SMD Packages
3D Packages shareable as STEP/OBJ file
Library.io site
Automatic Label with NET and BUS
V 9.1
PinBreakout
MOVE 45/90° maintaining angles
CAM Processor improvements
New Directory structure
HIG Interface
3D Packages editable inside EAGLE
V 9.3
Fanout – Allow violation mode
Selection filter
FUSIONTEAM publishing
Spice Sim Window non modal
CAM Processor improvements
Package Generator for 2D and 3D
V 9.5
Managed Folders
Groups
Design Manager in schematic
Filter
LOCK command extension
CAM Processor – Pick&Place
Library – Remove objects/variants

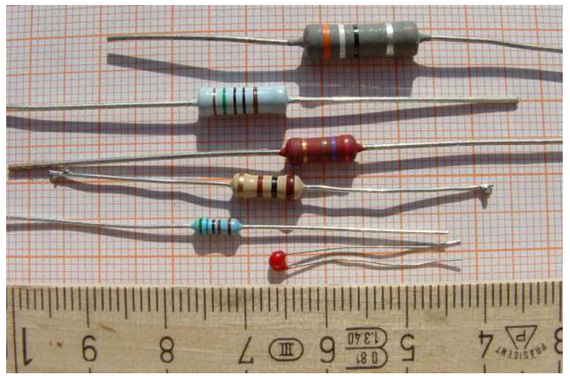
Electronics?

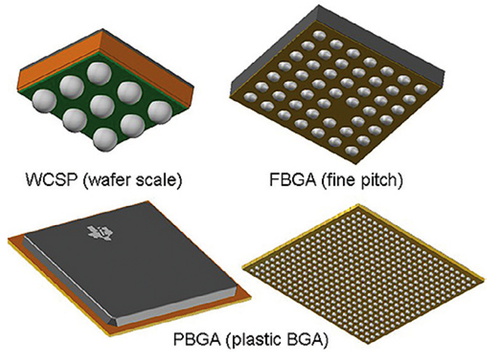
Electronic Components



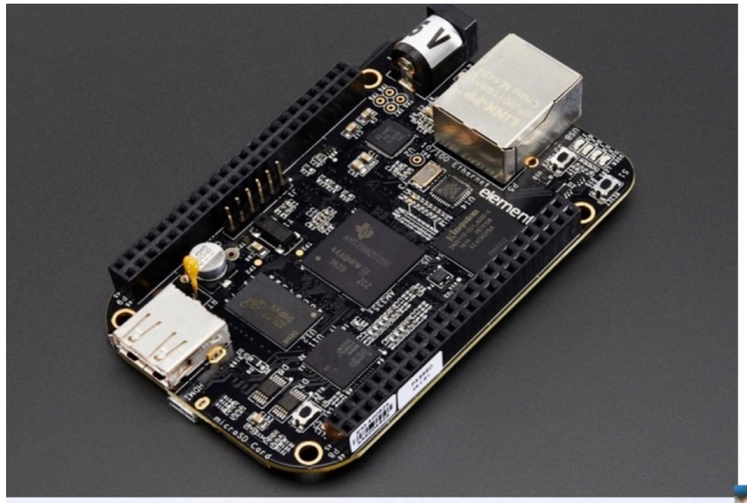


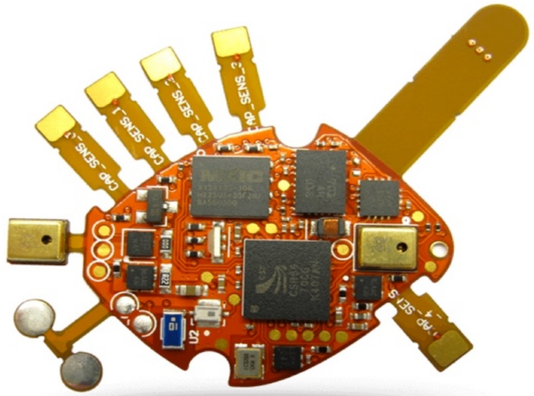


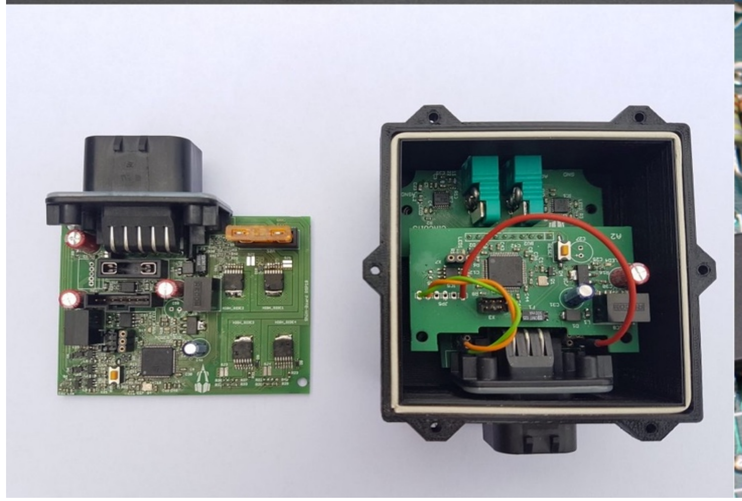


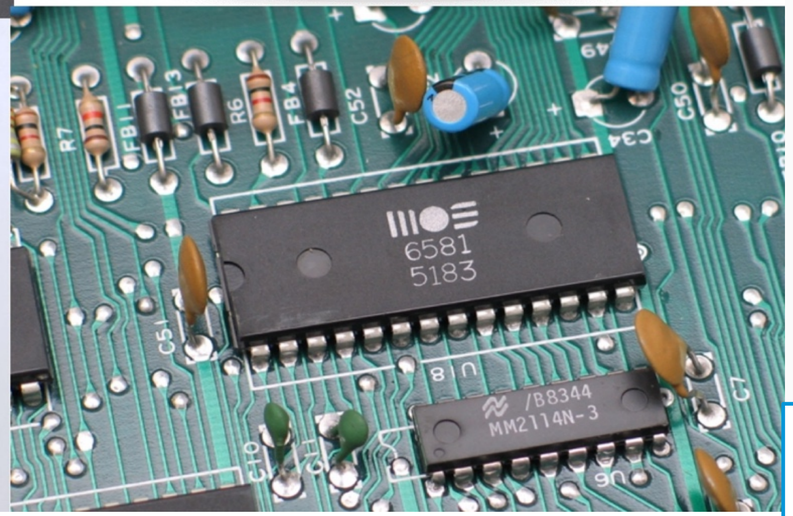


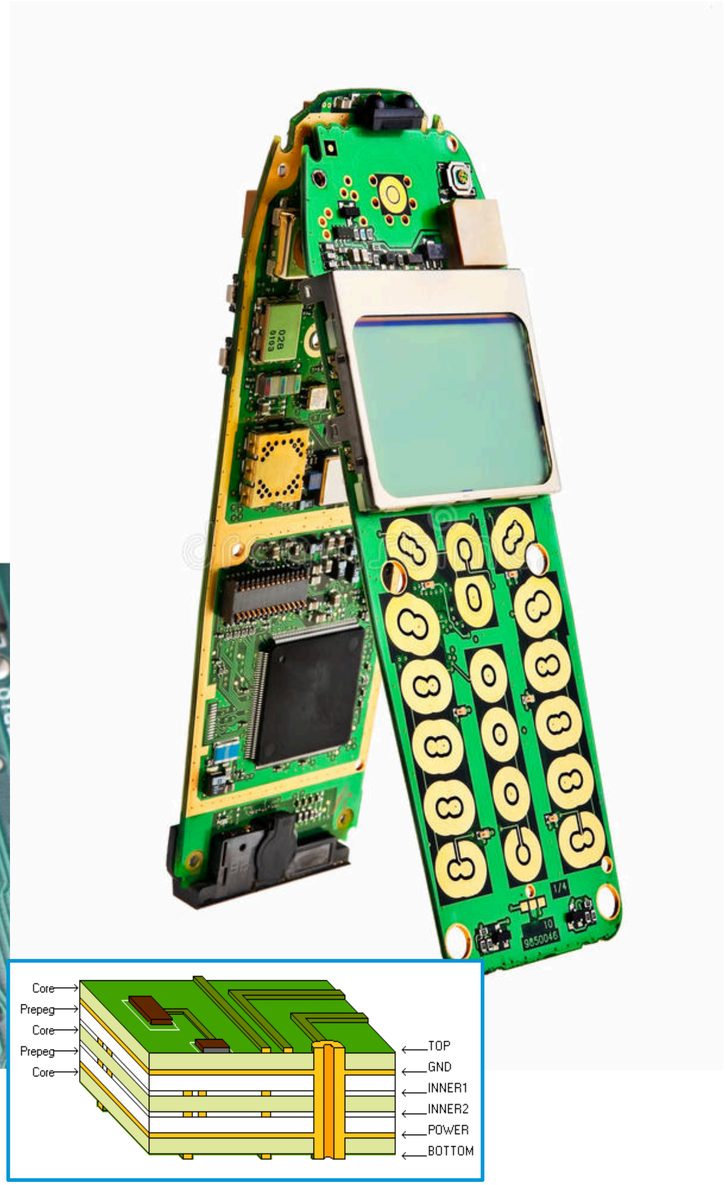
Printed Circuit Boards



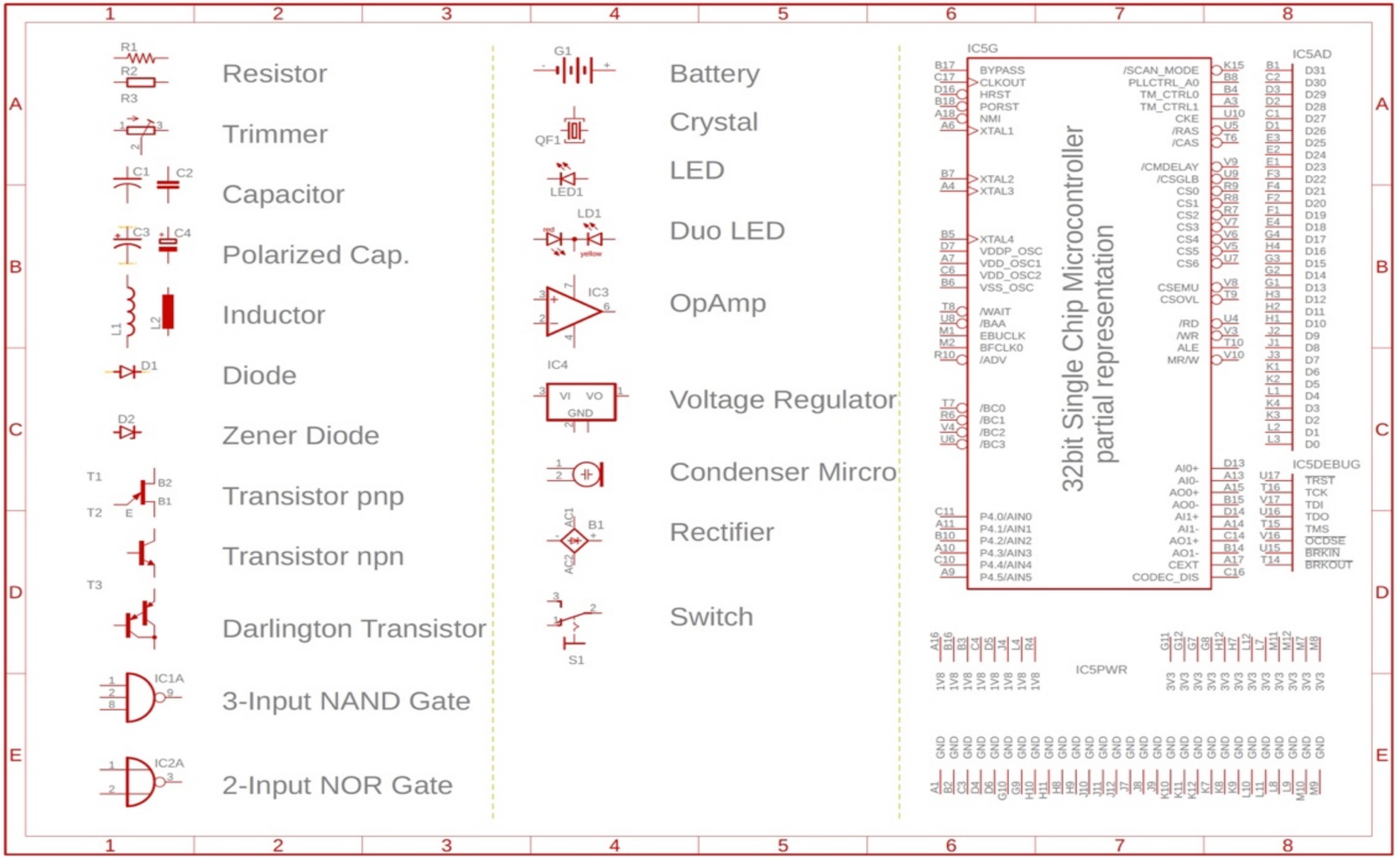






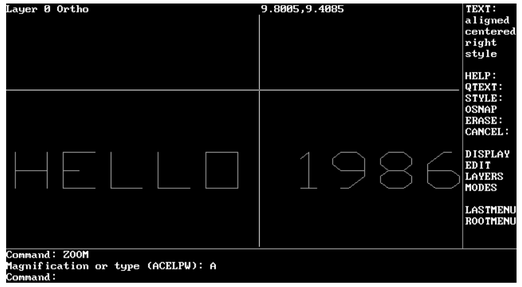


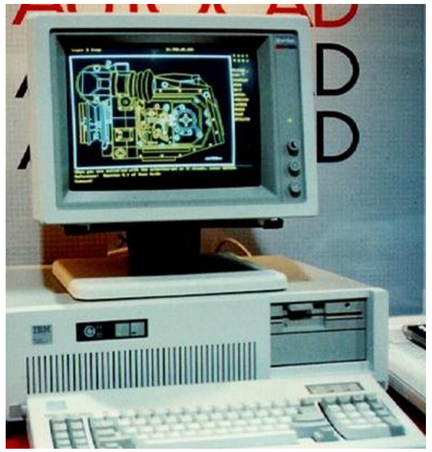
Schematic Representation of Components

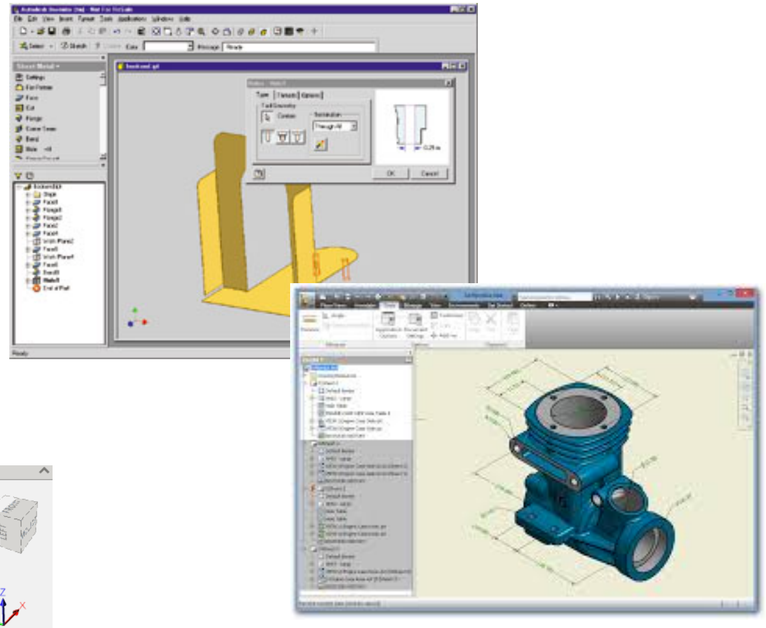


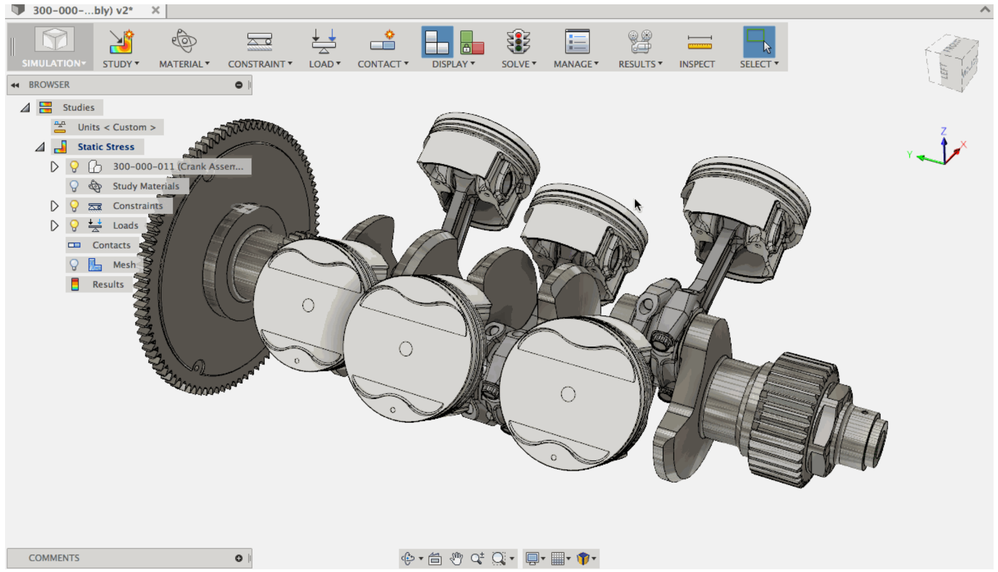
A Bit of History…

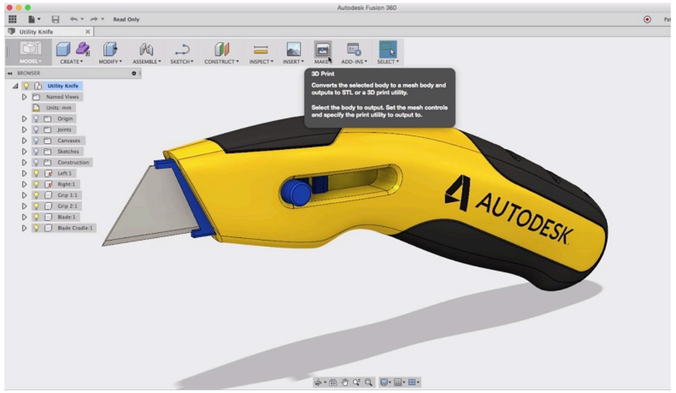
Mechanical Design



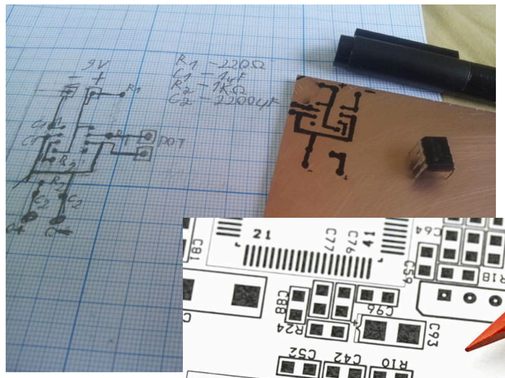


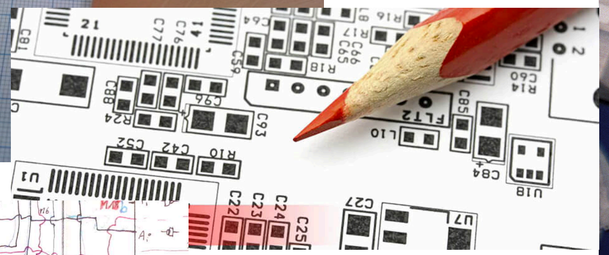


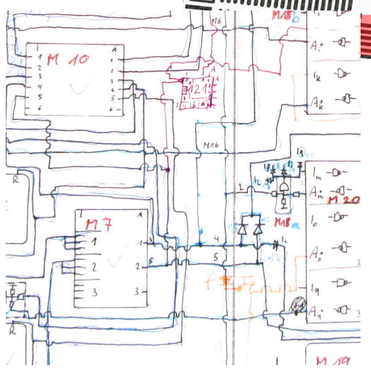


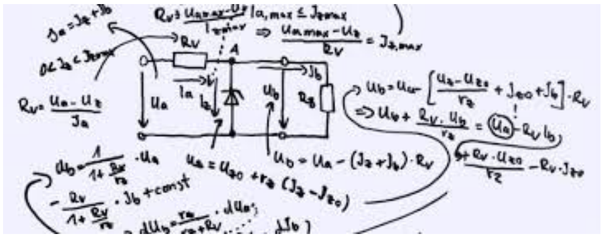


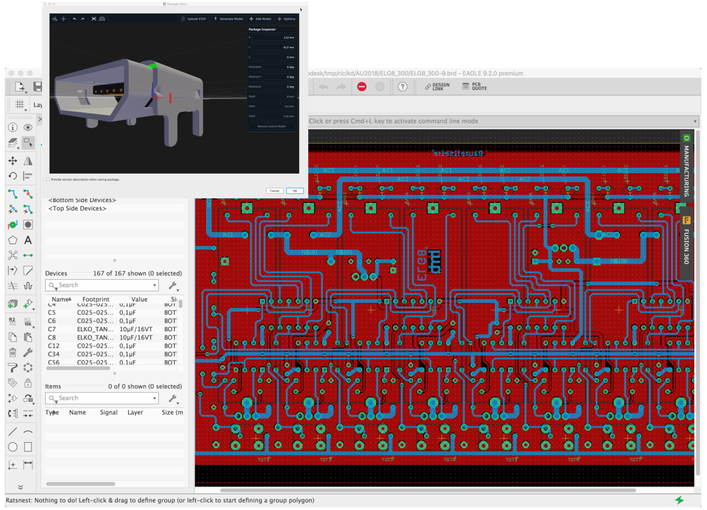
Electronics Design



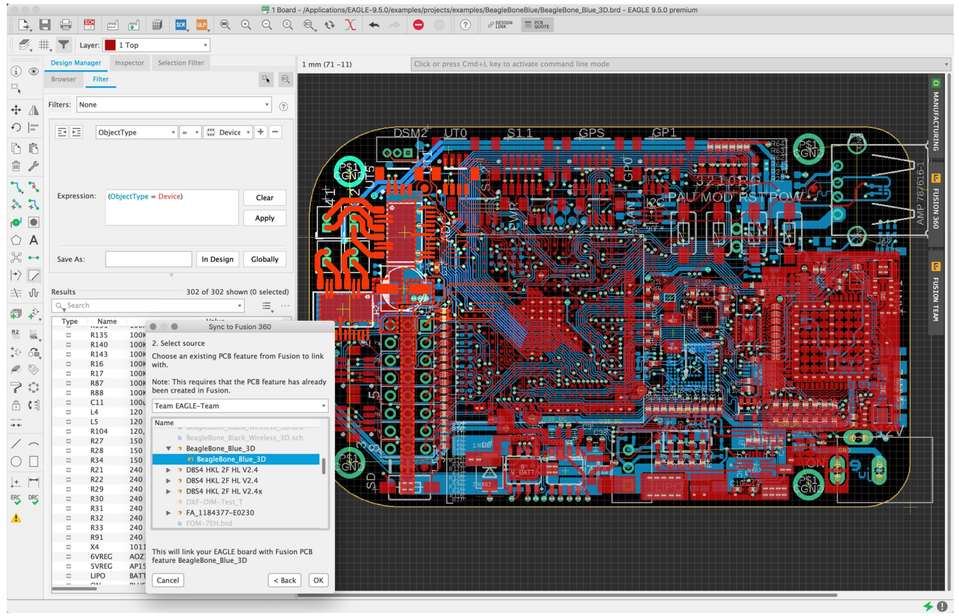


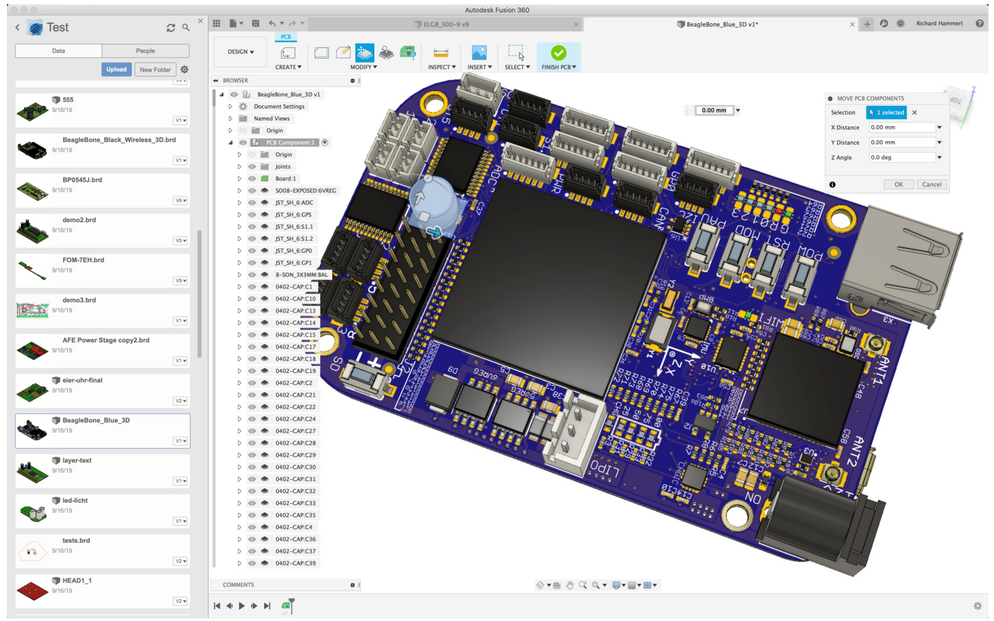






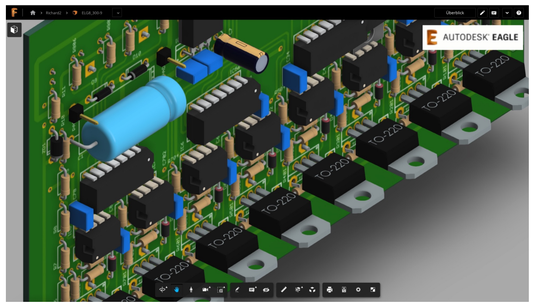
Product Development MCAD + ECAD

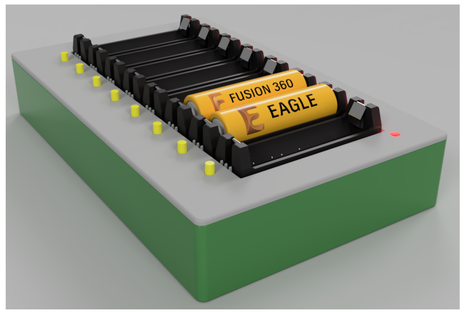


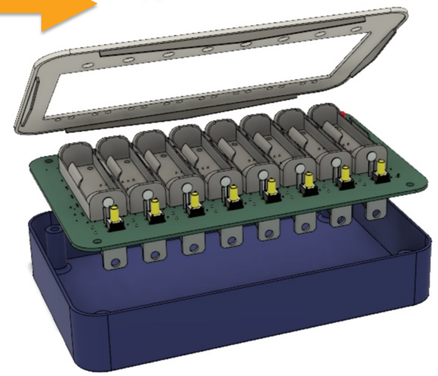


AUTODESK® EAGLE

AUTODESK® FUSION 360™





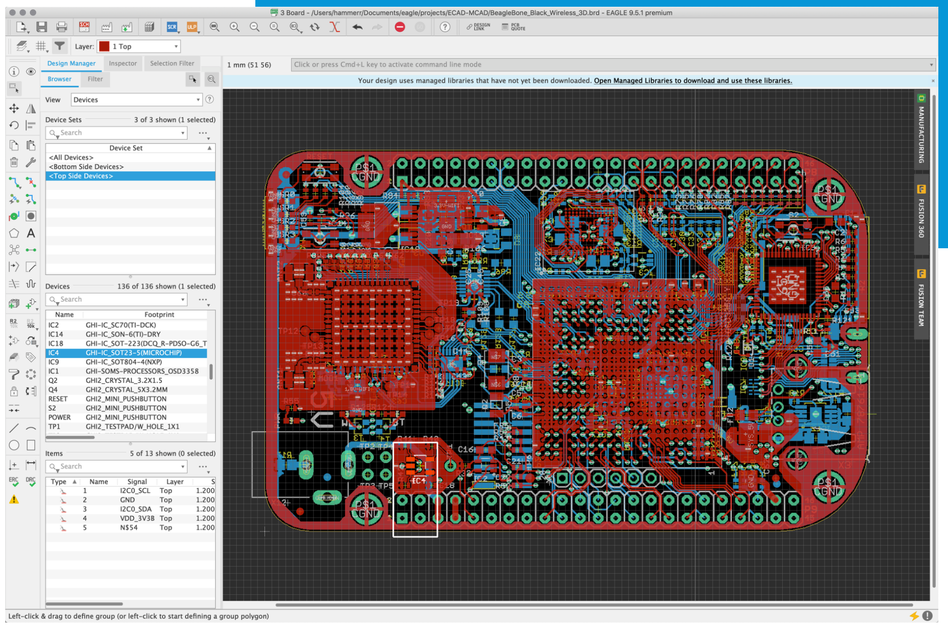


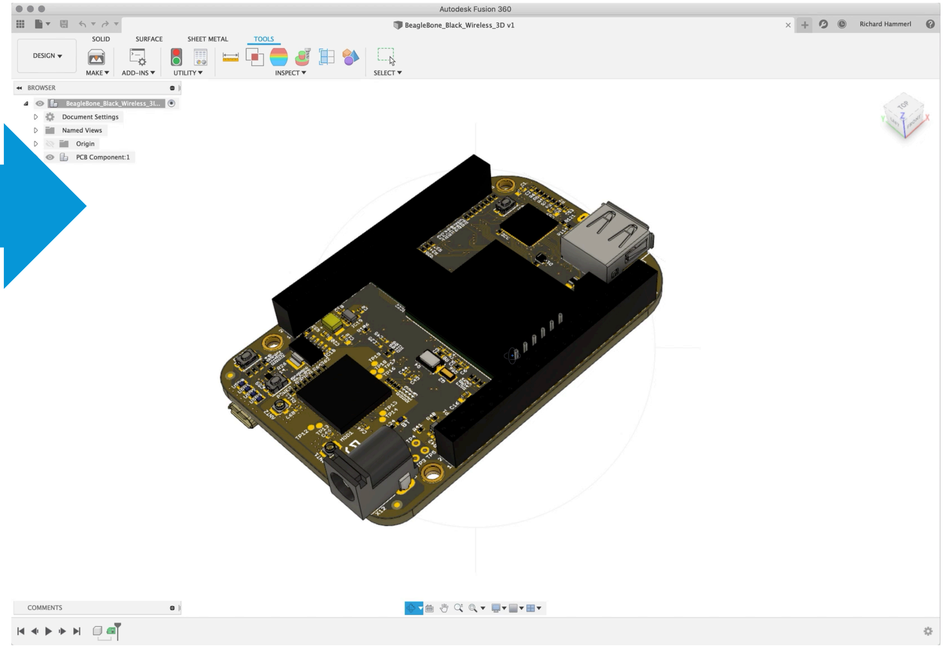
Product Development MCAD + ECAD

AUTODESK® EAGLE

DATA CENTERED

AUTODESK® FUSION 360™





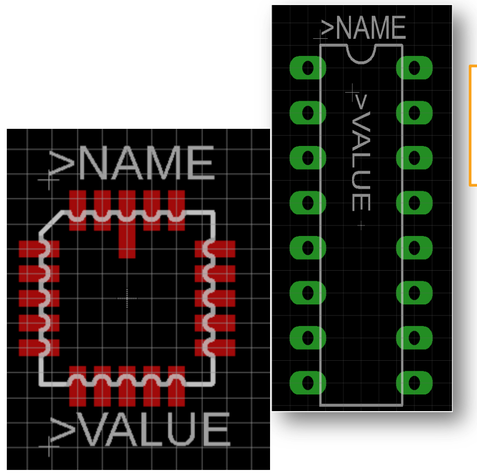
Fusion 360

AUTODESK® **FUSION 360**™

https://www.autodesk.com/products/fusion-360/features

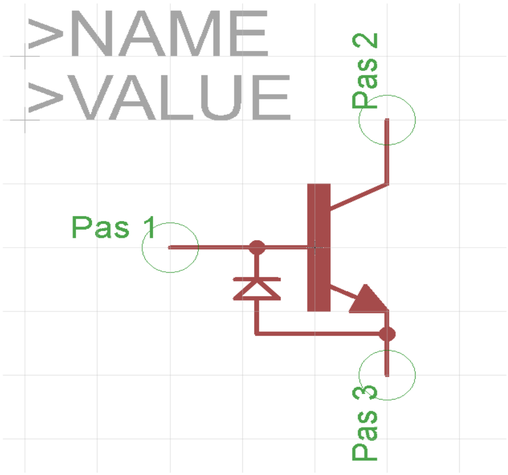
EAGLE

Component Libraries



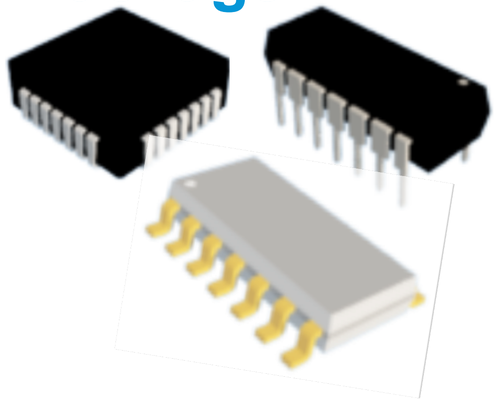
Footprint

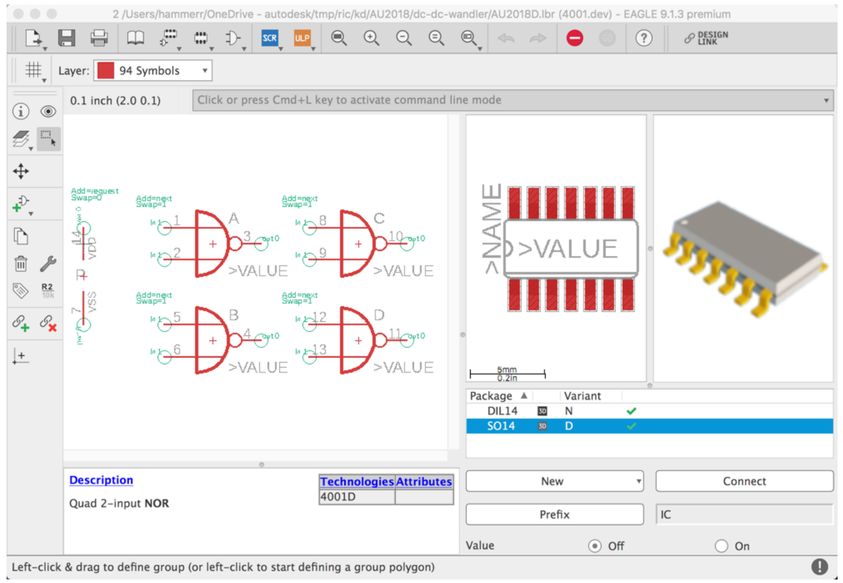
Symbol



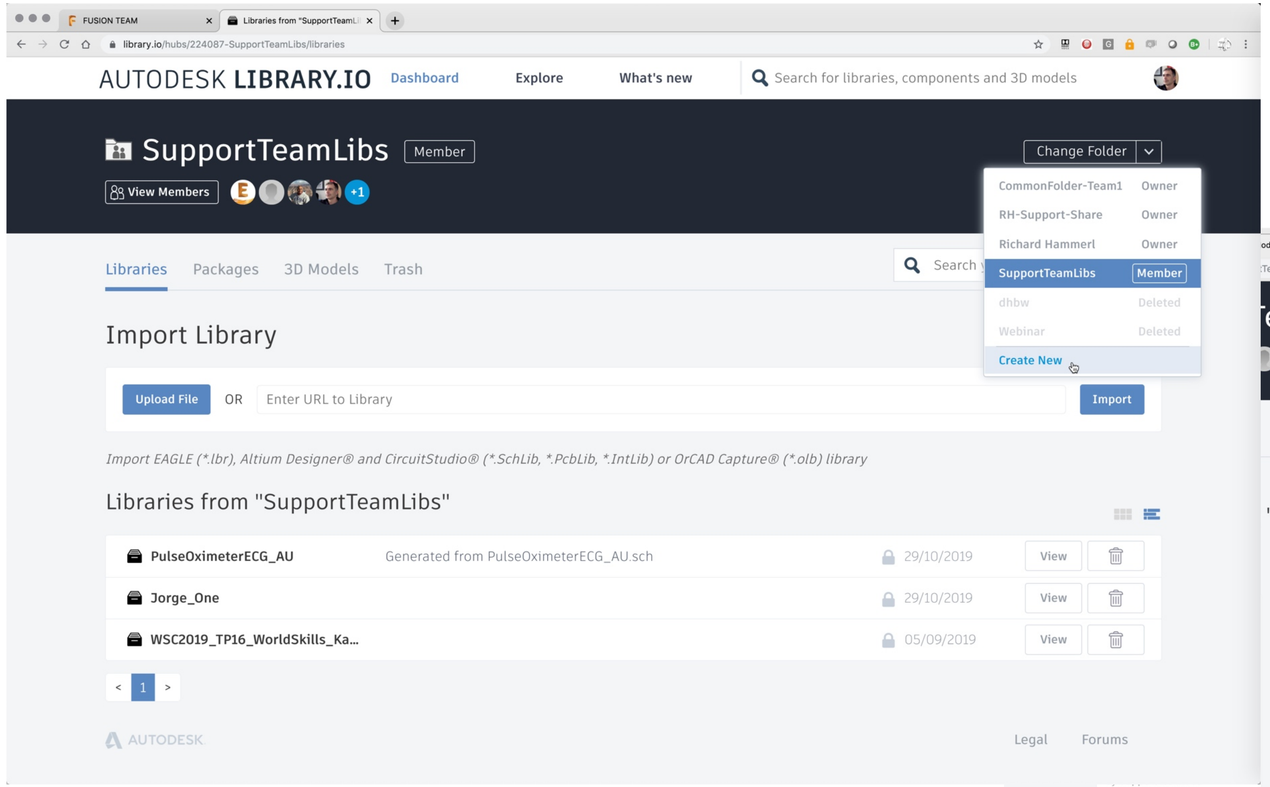
Device

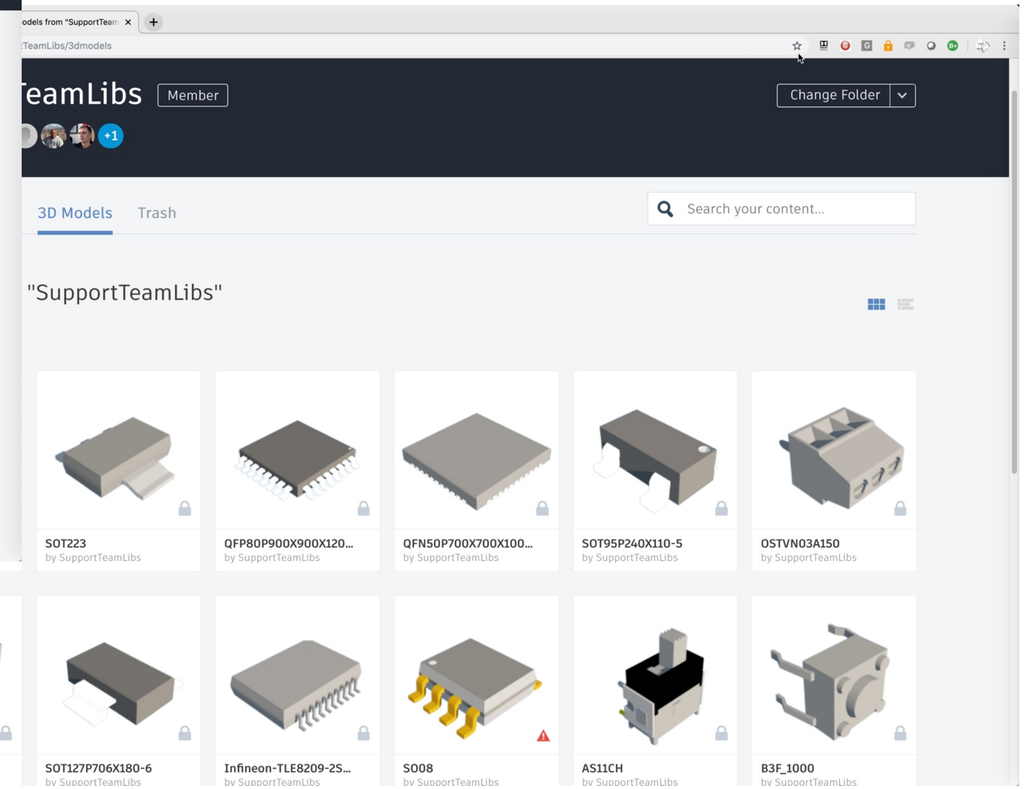
3D Package





Shared Library Folders



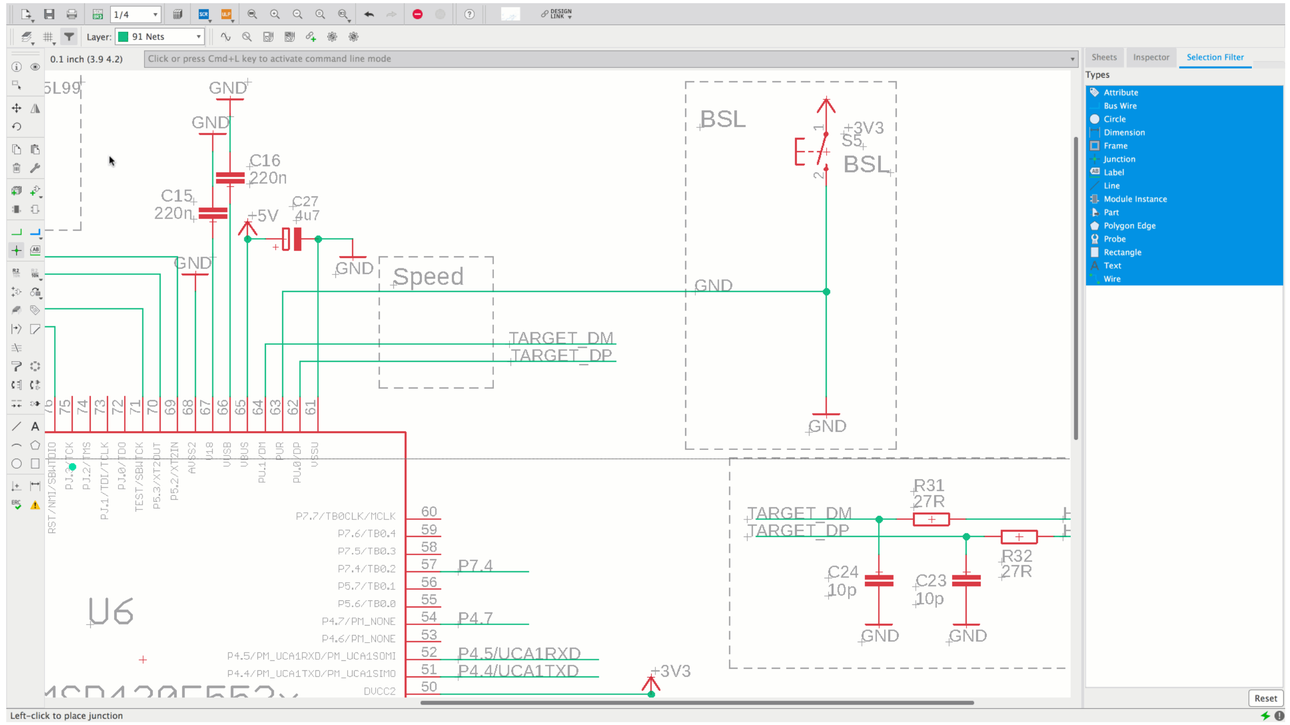


- Libraries can be shared with team members
- Access from anywhere and any computer
- Up to 16 shared folders

Schematic Design

An idea is brought to "paper".

- Placing components (symbols) and connecting the connection points (pins) with nets.
- Use of Design Blocks for recurring parts of a circuit.
- EAGLE provides component libraries with the installation package.
  Additional libraries online.
- Up to 999 schematic sheets per schematic.

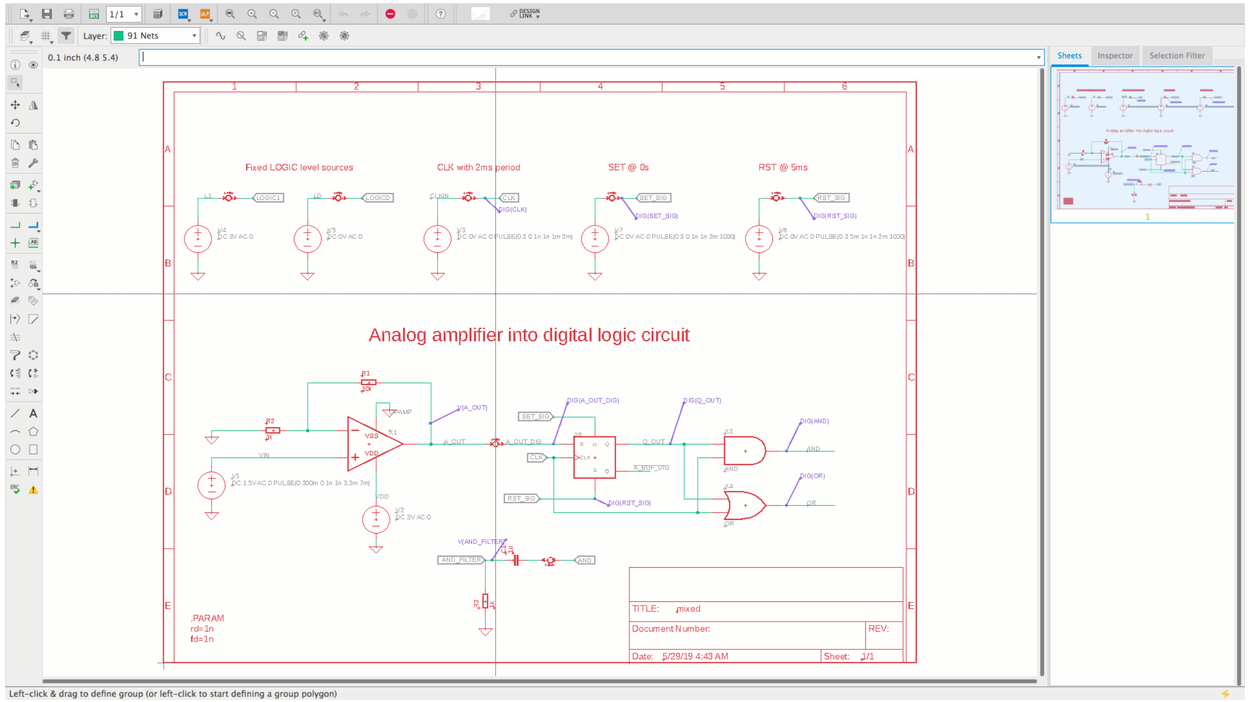


Schematic Editor

SPICE Simulation

Is my circuit working as expected ?

- SPICE (**S**imulation **P**rogram with **I**ntegrated **C**ircuit **E**mphasis) is a program for the simulation of analog, digital and mixed electronic circuits.
- EAGLE uses the ngspice simulator to do this.
- Ability to add SPICE models to components.
- Output of plots and display of simulation values directly in the schematic

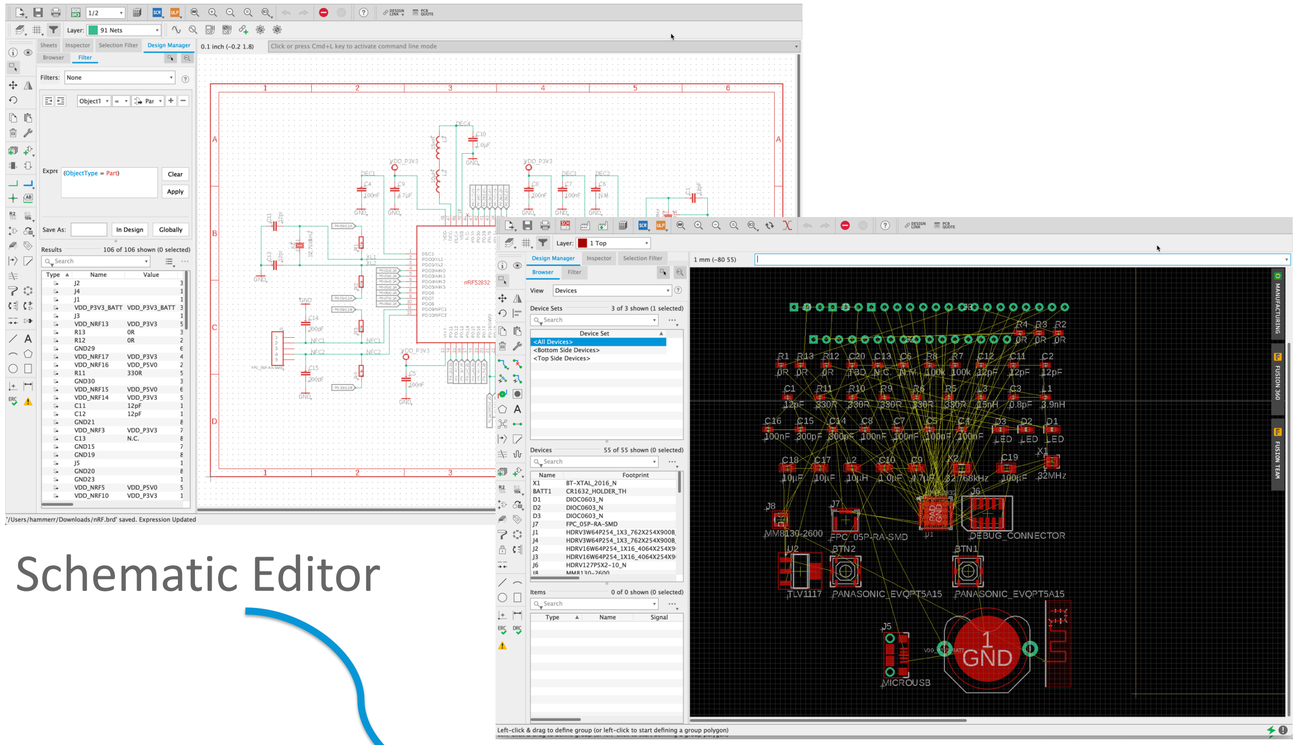


SPICE Simulation of the circuit

From Schematic to Layout

The idea becomes reality

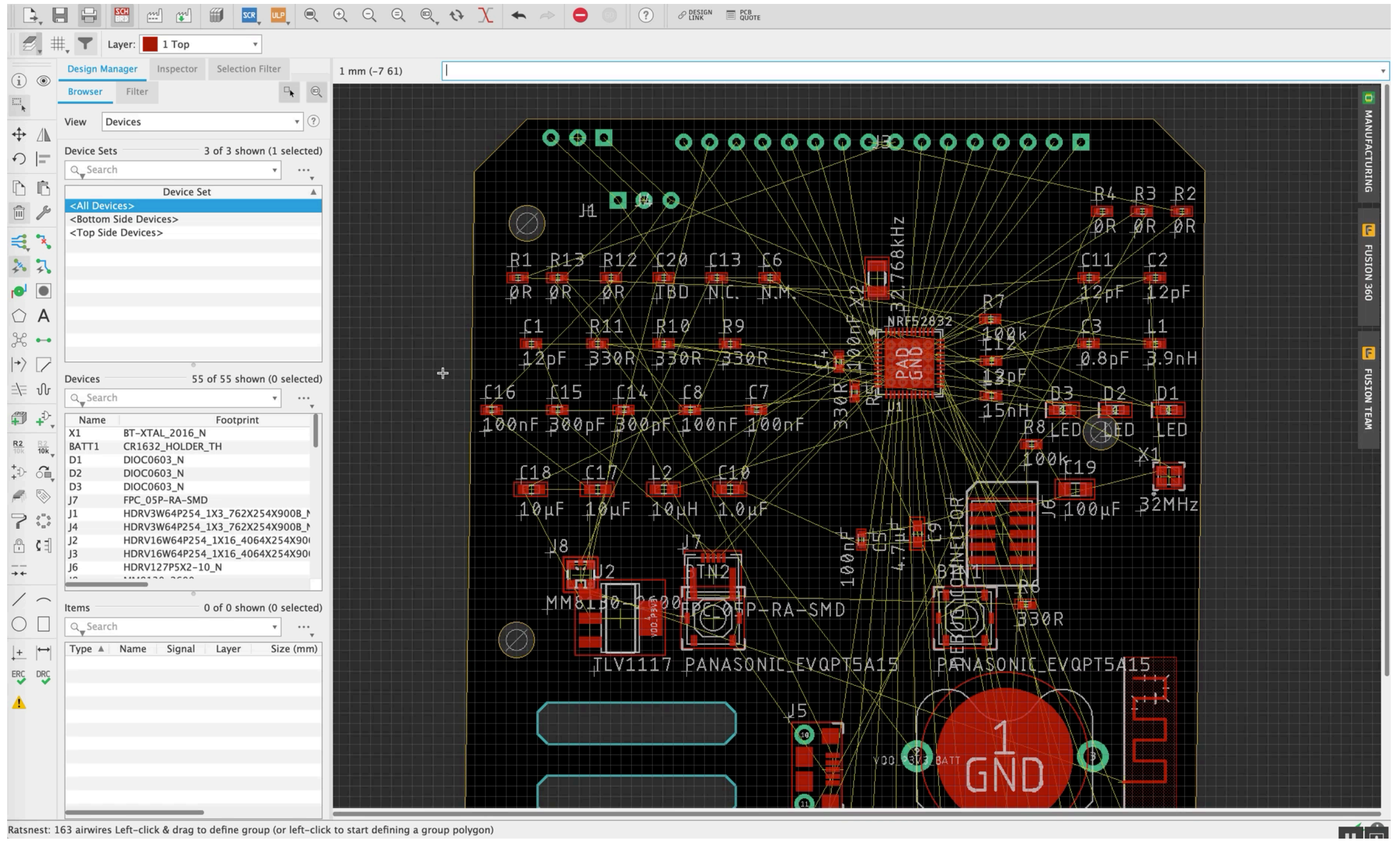
- With one click you can create a layout from the circuit diagram
- There is a Forward-/Back-Annotation between schematic and layout: Every change in the schematic is immediately transferred to the layout and vice versa.
- The schematic symbols have already been assigned to packages in the libraries.
- The nets from the schematic appear as signals (airwires) between the component soldering pads.



Layout Development I

This is where the printed circuit board is designed - the real product

- Defining the PCB geometry
  - Transfer from Fusion 360
  - Import from e.g. DXF file
  - Create in EAGLE
- Definition of Design Rules
  - layer stackup
  - Production class with related minimum values
- Arrangement of the components
- Routing the traces
  - Manually
  - Obstacle Avoidance
  - QuickRoute
  - Push&Shove
  - Smooth signals
  - Automatic routing

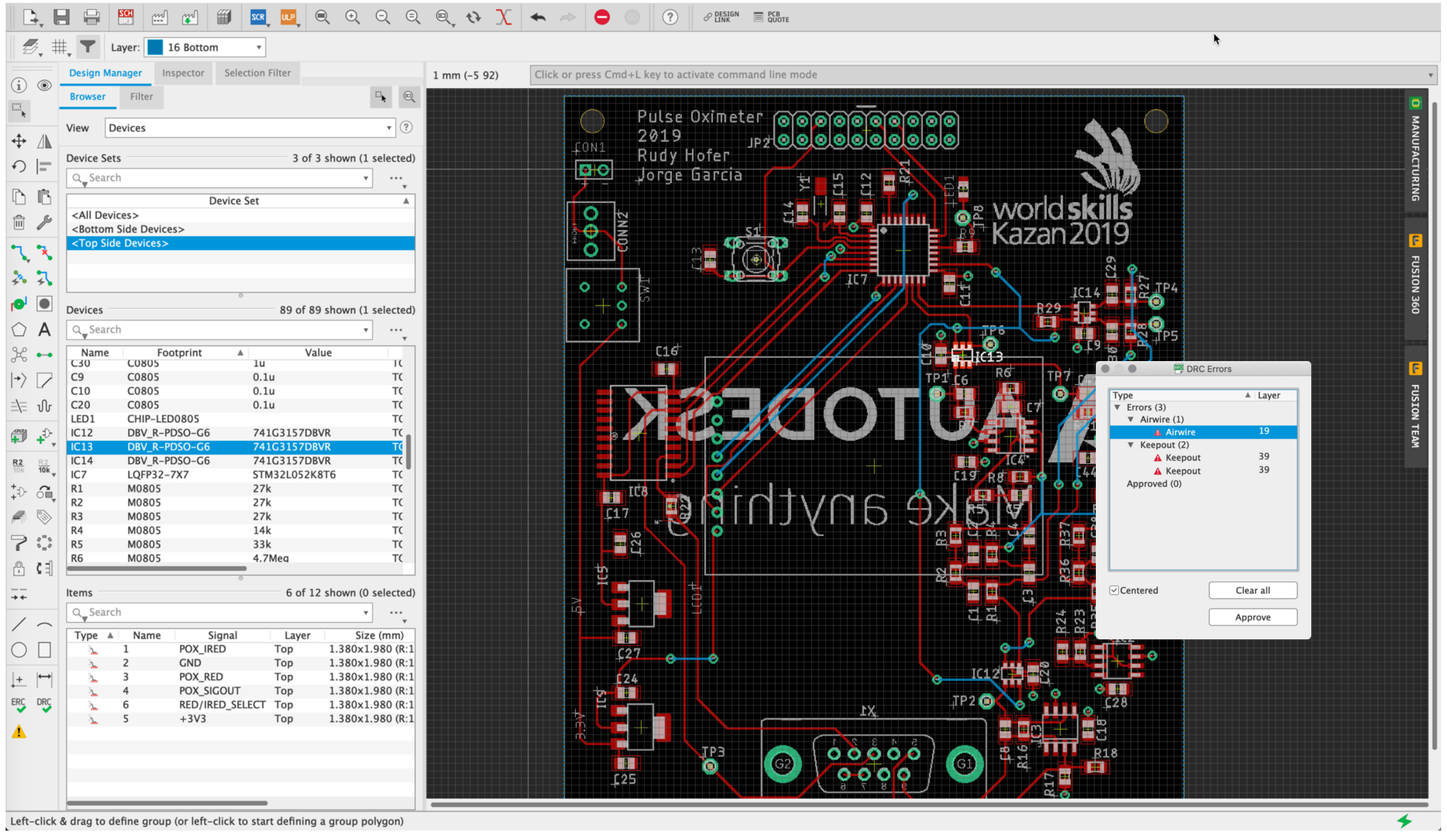


Setting the Design Rules and routing the first traces

Layout Development II

Ready for data output and production

- Final Design Rule Check:
  - All rules met?
  - All signals routed?
  - No errors in the design?
- Is the PCB‘s geometry correct?
- Positions of the components?
- Final Push and Pull for synchronisation with the mechanical design.

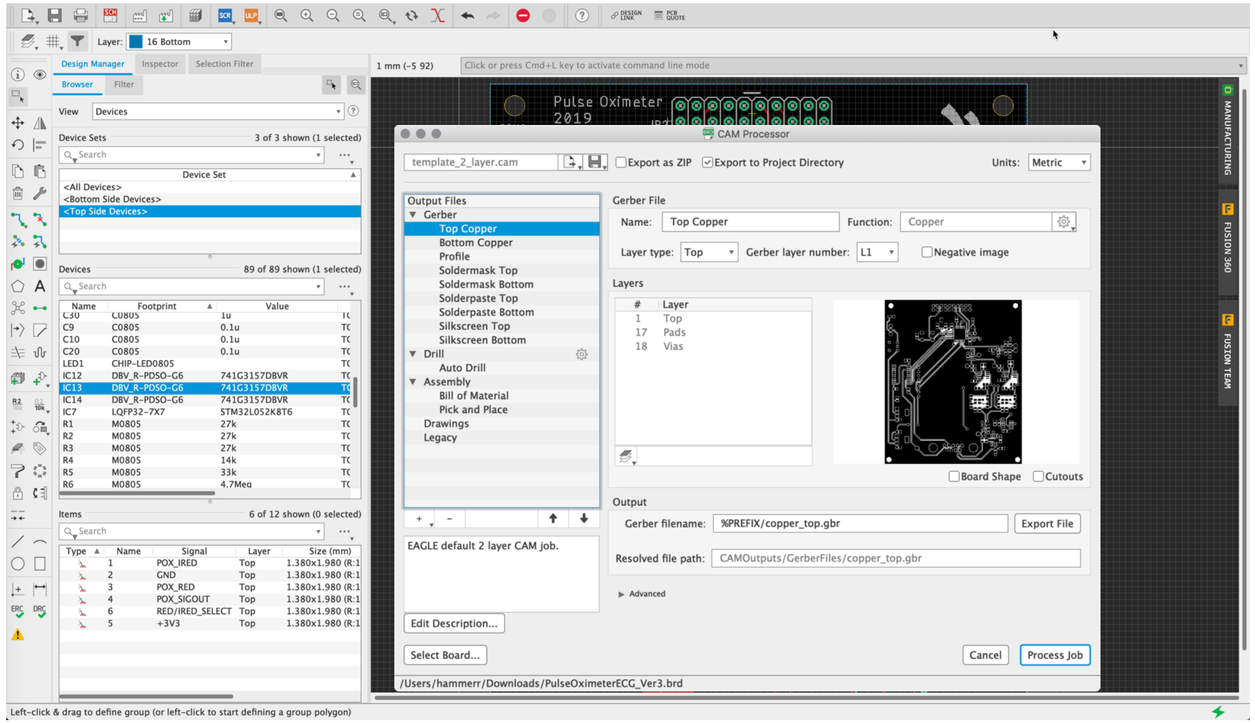


Design Rule Check indicates possible problems

Data Output for Manufacturing

Creation of production data

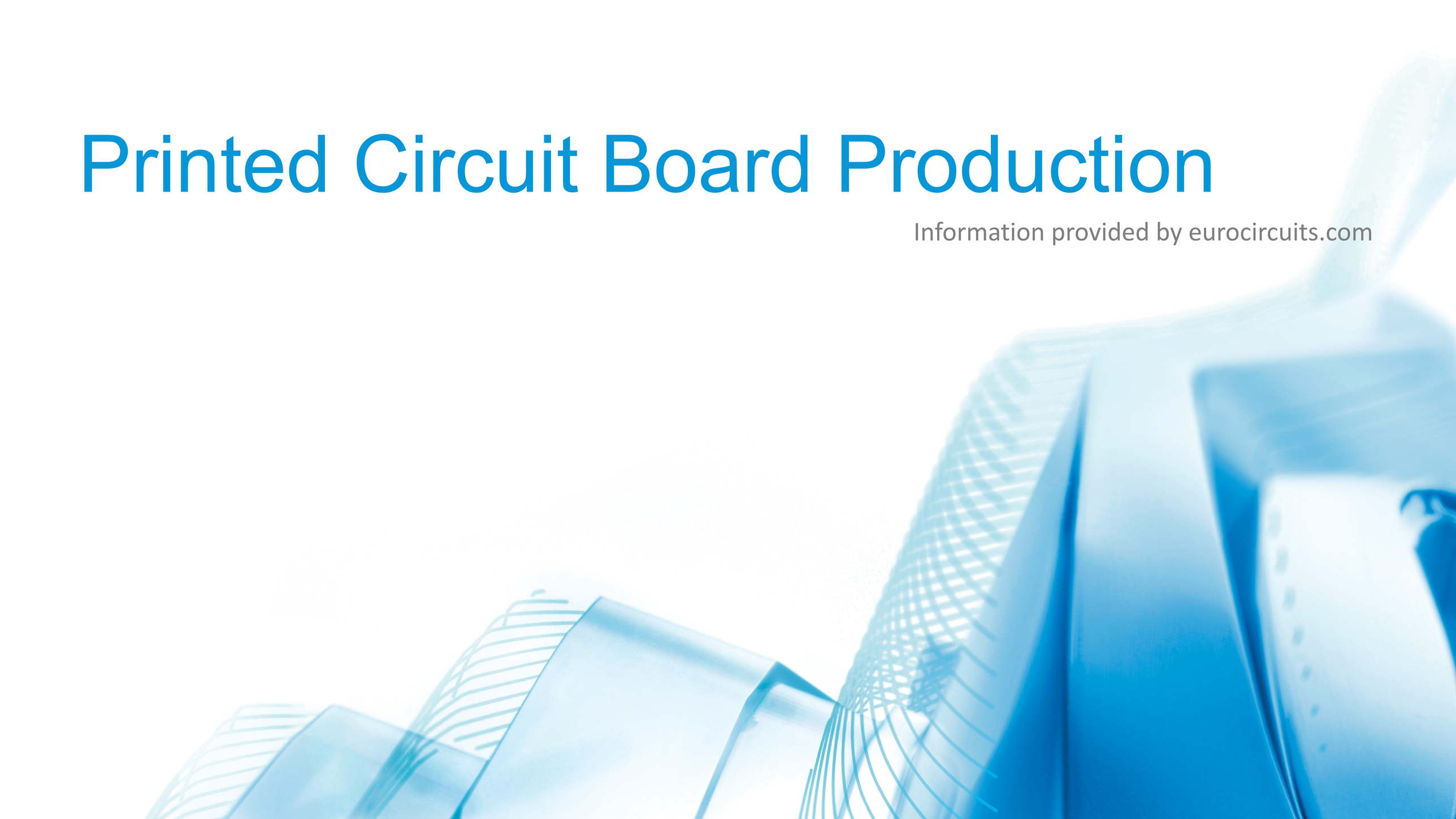
- Data output in Gerber format and drilling data in Excellon format
- Generating a bill of material
- Creating Pick & Place data for an assembly service
- Alternatively you send the layout file to the board manufacturer. They will generate all the necessary data for you.



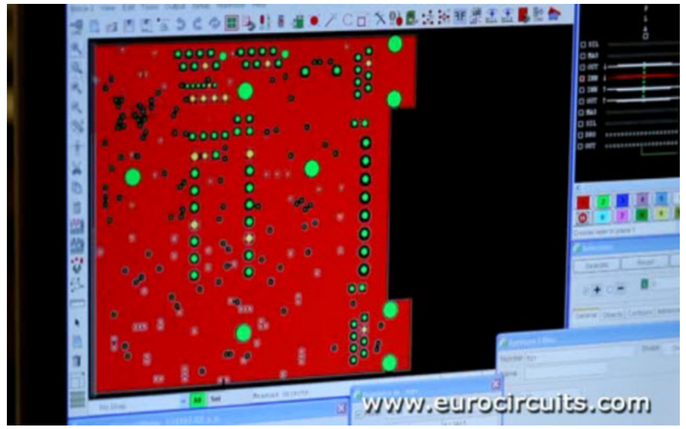
CAM Processor data output

Printed Circuit Board Production

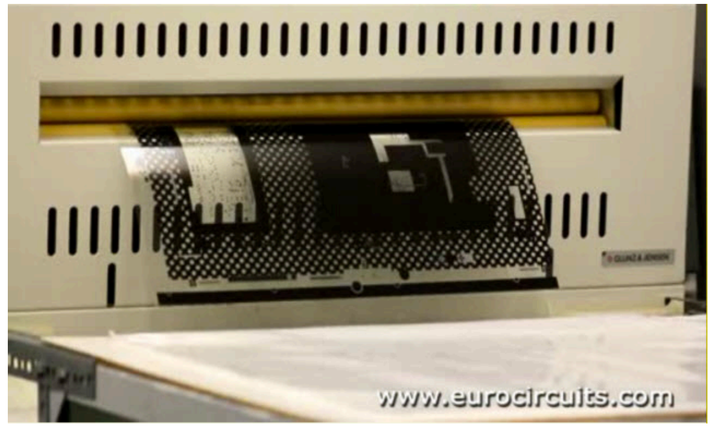
Information provided by eurocircuits.com



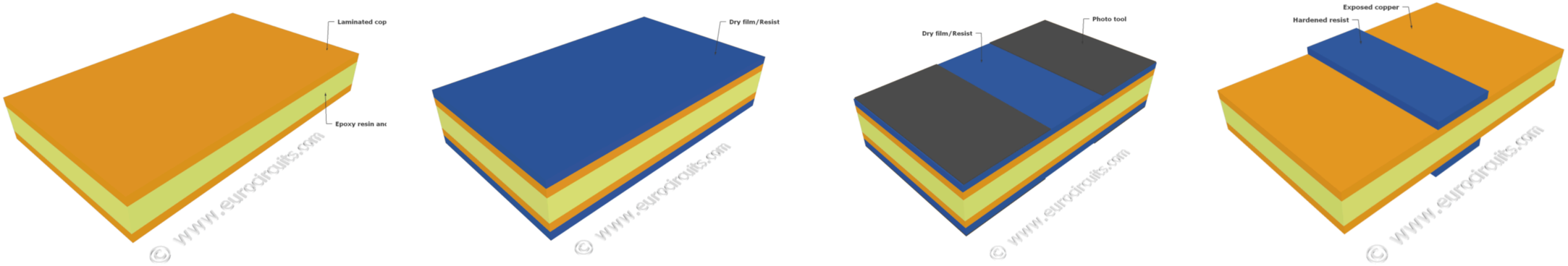
PCB Production 1-3



1. Data preparation

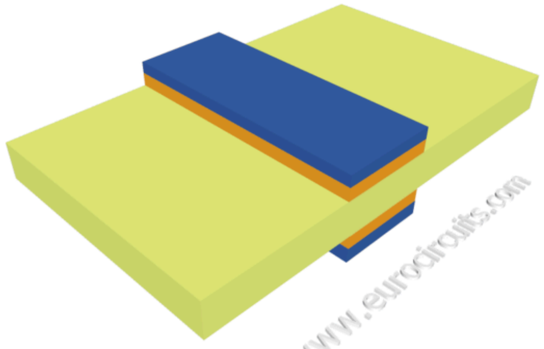


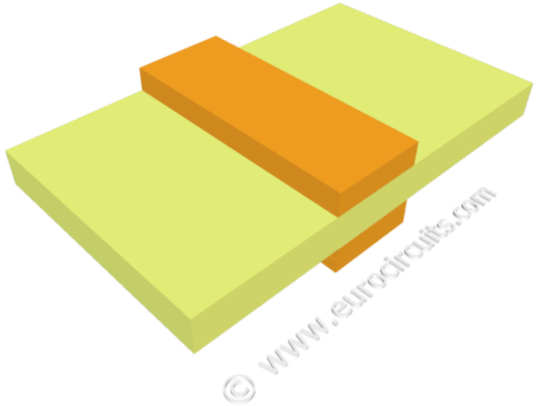
2. Image transfer



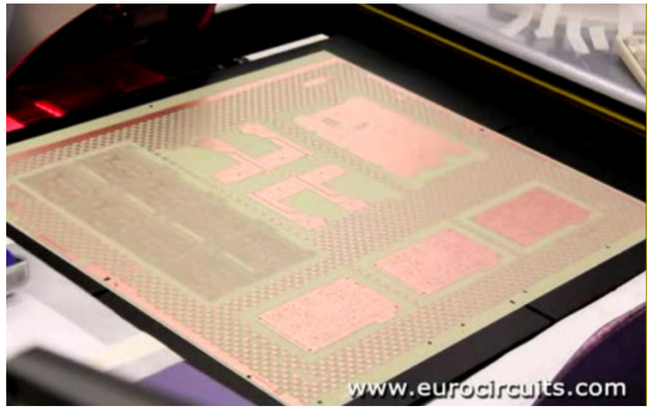
3. Inner layer imaging for multilayer board

PCB Production 4-6

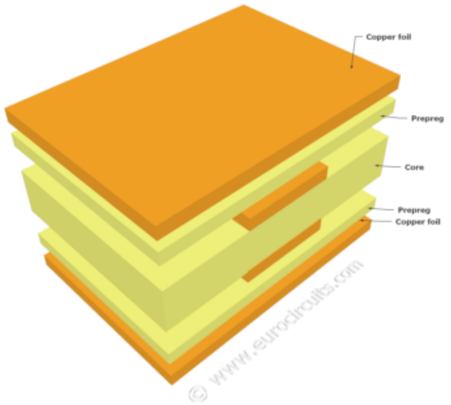


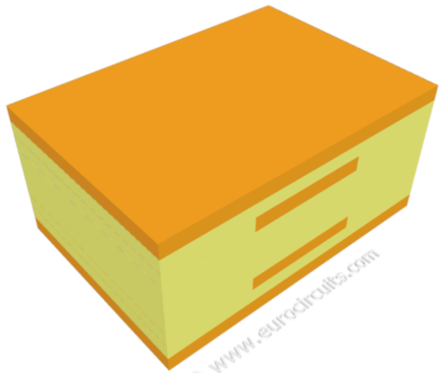


4. Etching inner layers



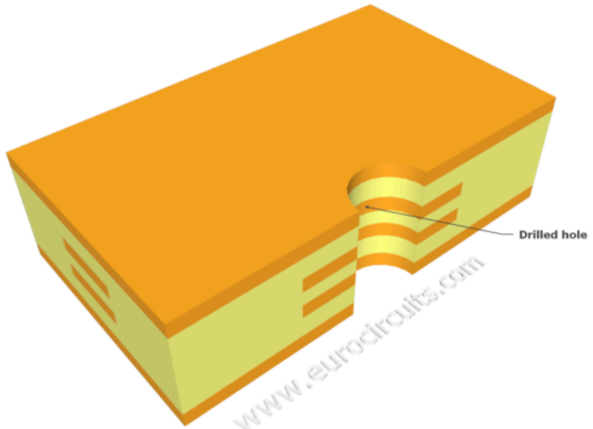
5. AOI –Automatic Optical Inspection



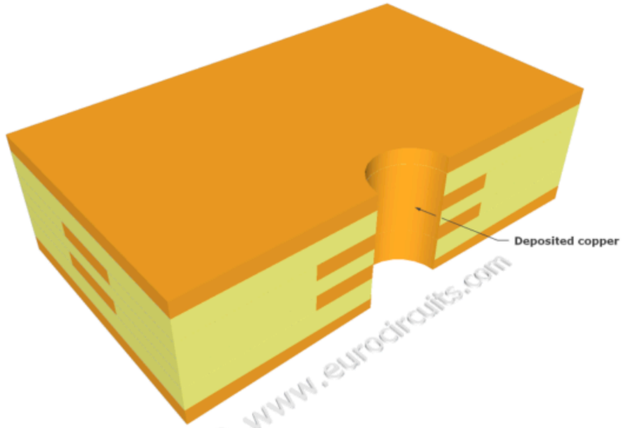


6. Lay up and bond (outer layers)

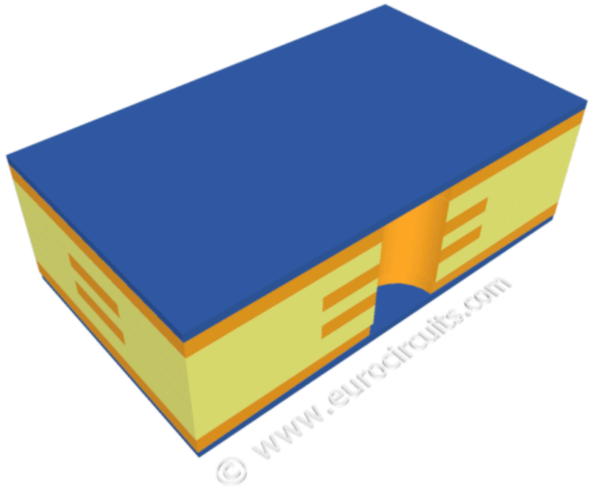
PCB Production 7-9



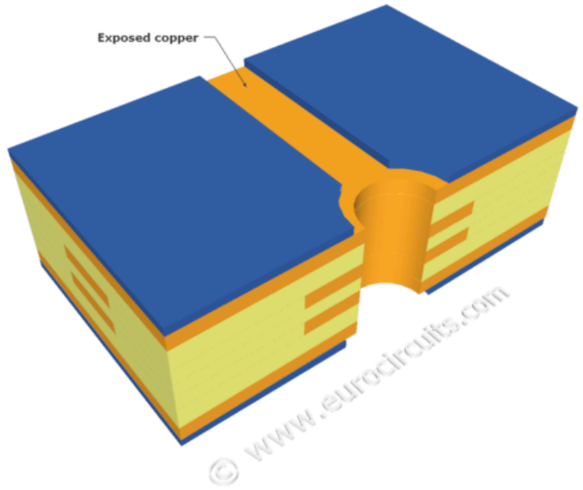
7. Drilling the holes



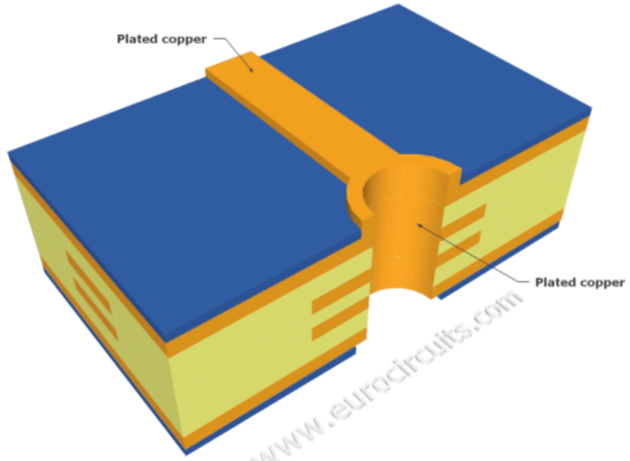
8. Electroless copper deposition

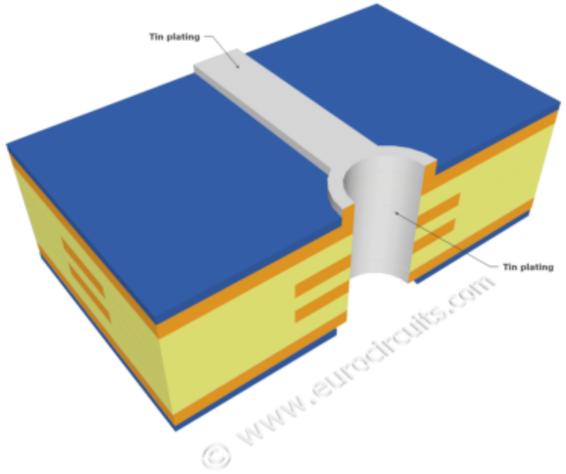


9. Imaging the outer layers

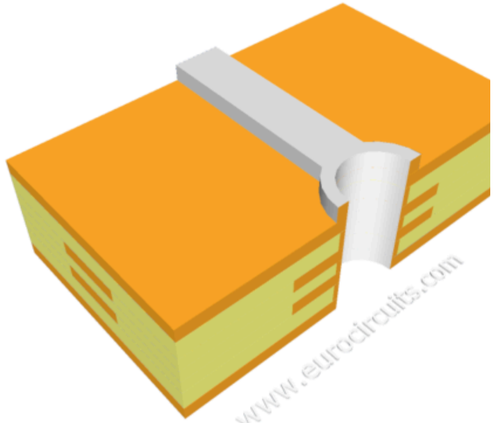


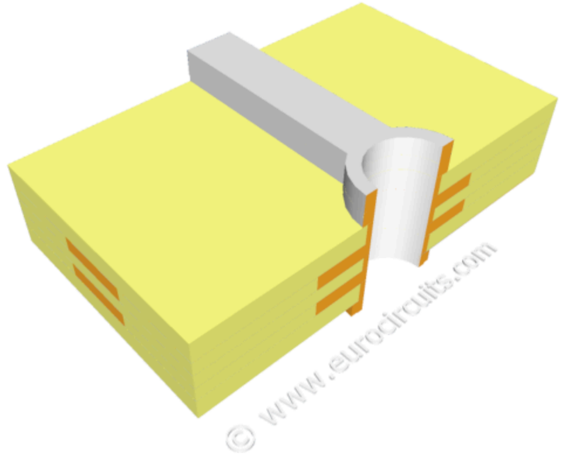
PCB Production 10-11

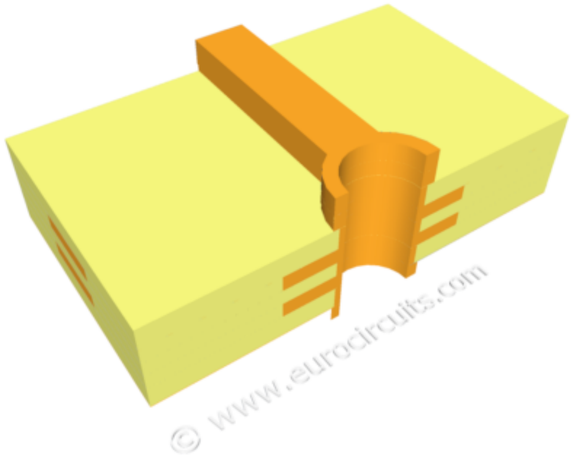




10. Plating

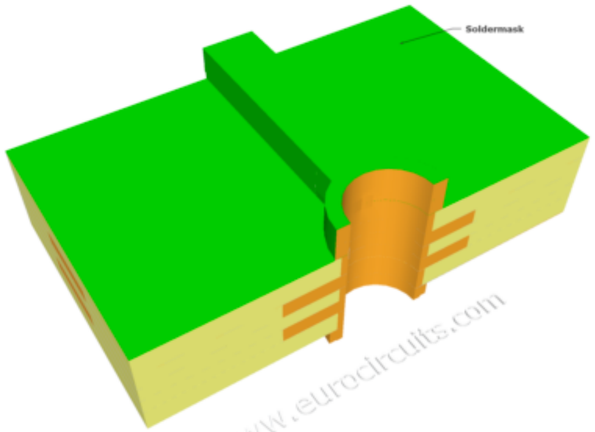




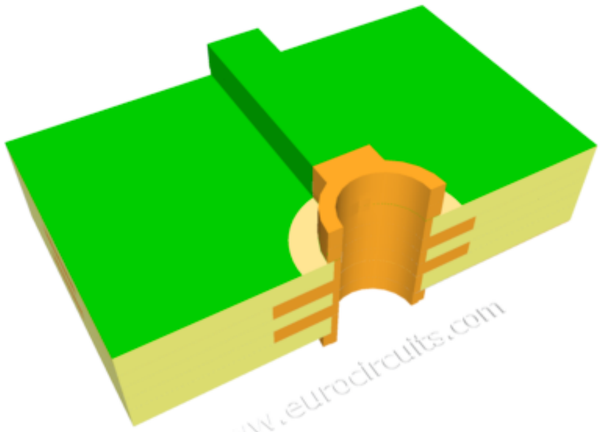


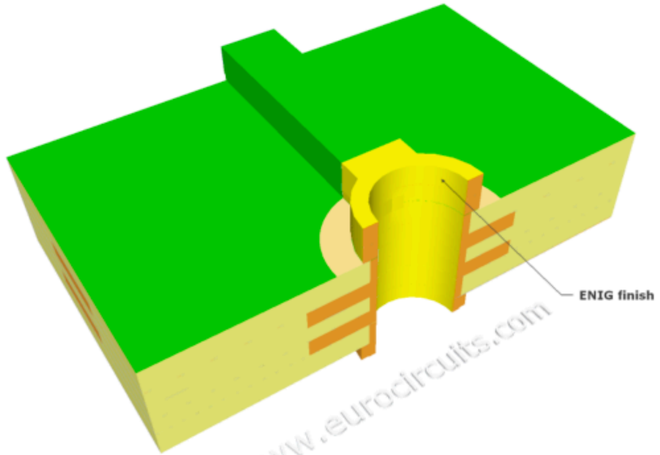
11. Etch outer layers

PCB Production 12-14

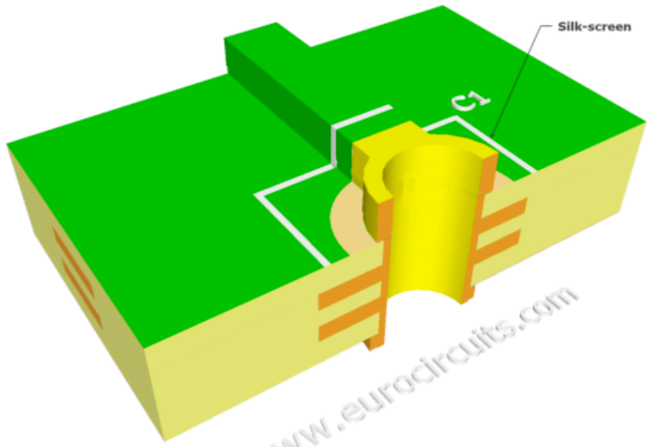


12. Apply solder mask



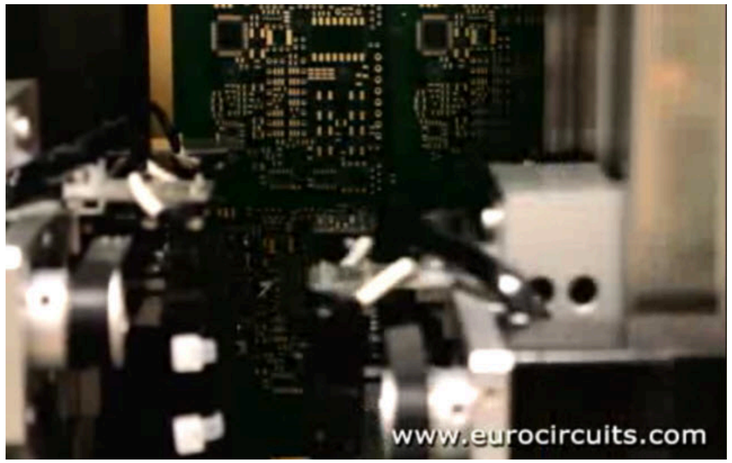


13. Surface finish

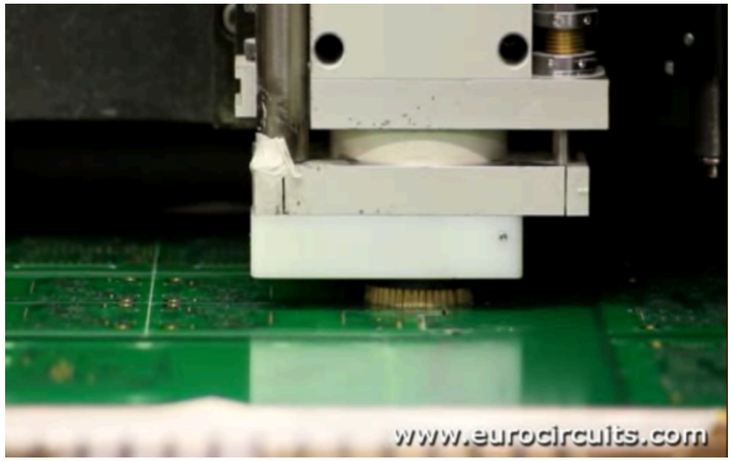


14. Silk screen

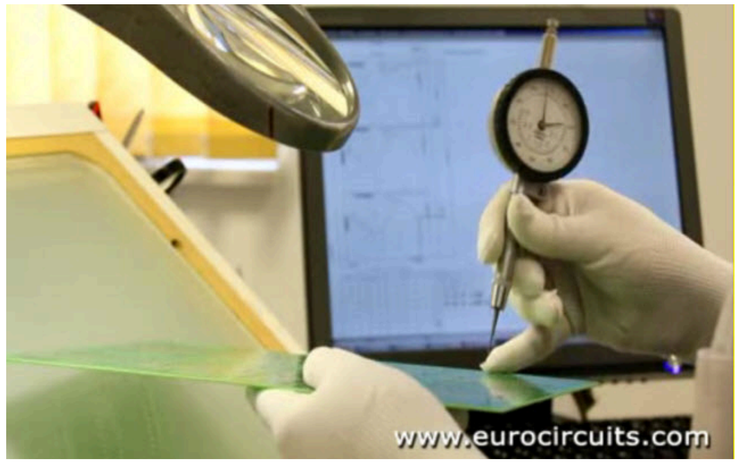
PCB Production 15-17



15. Electrical test

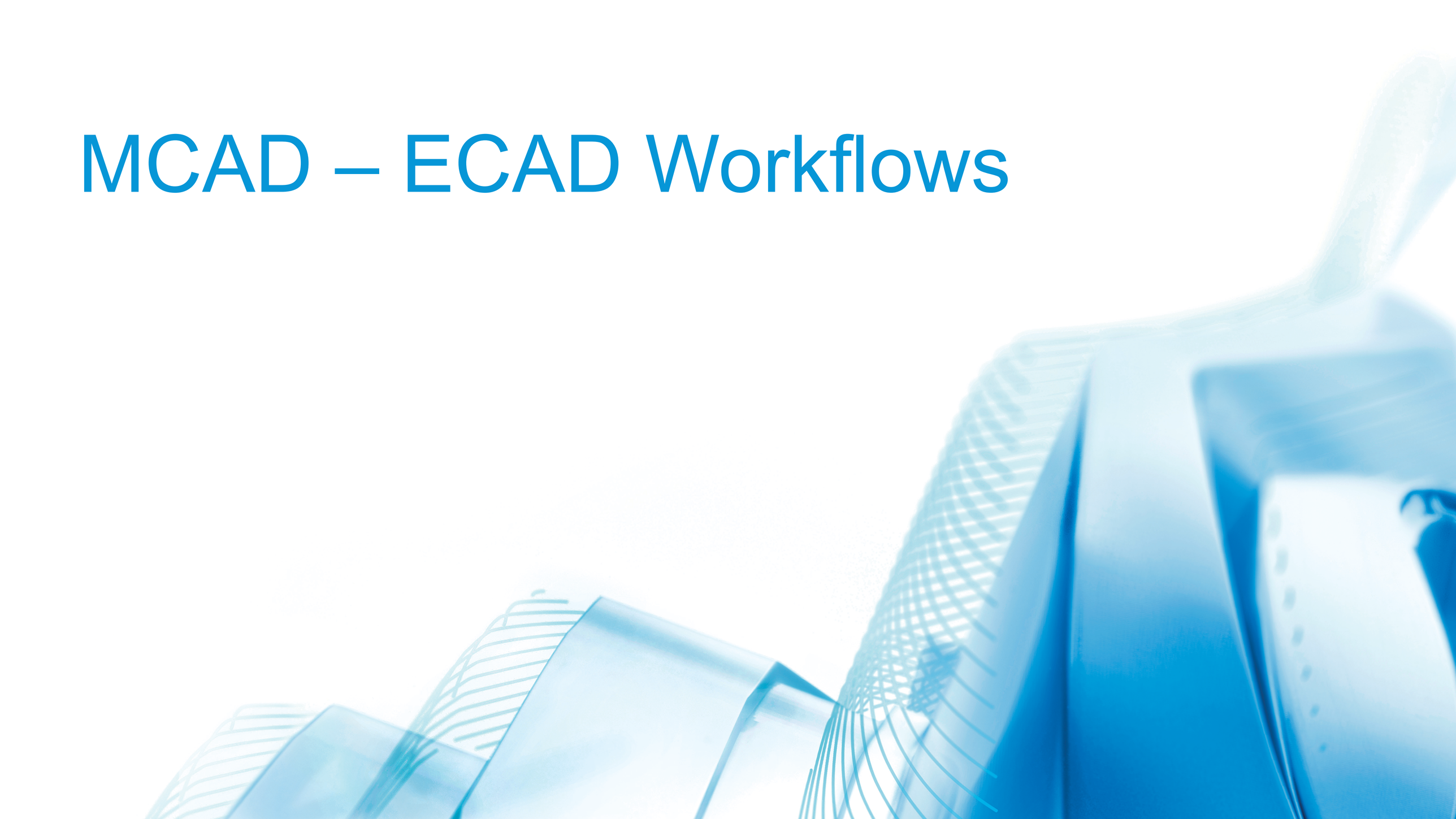


16. Profiling, V-cut scoring

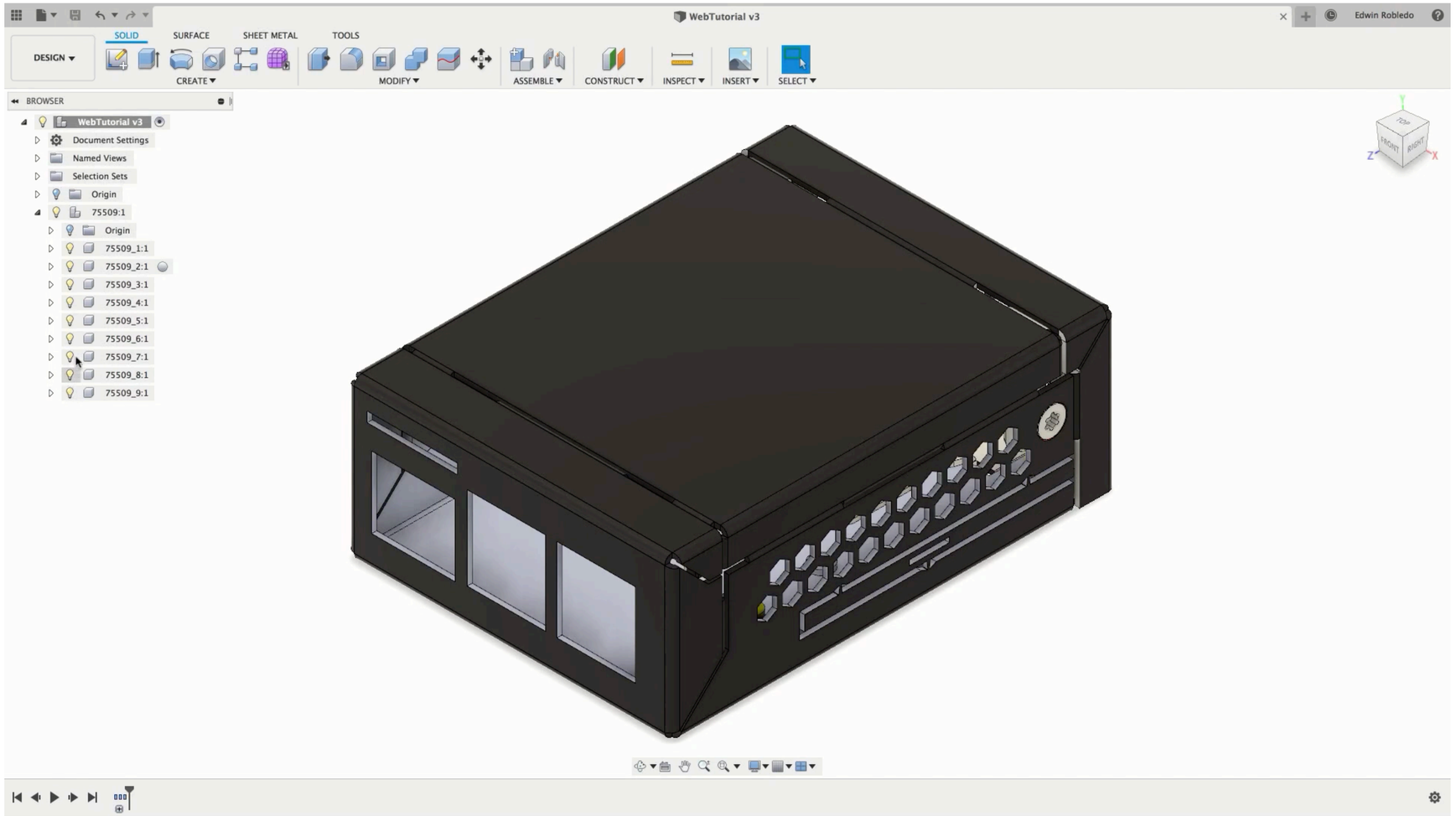


17. Final inspection

MCAD – ECAD Workflows



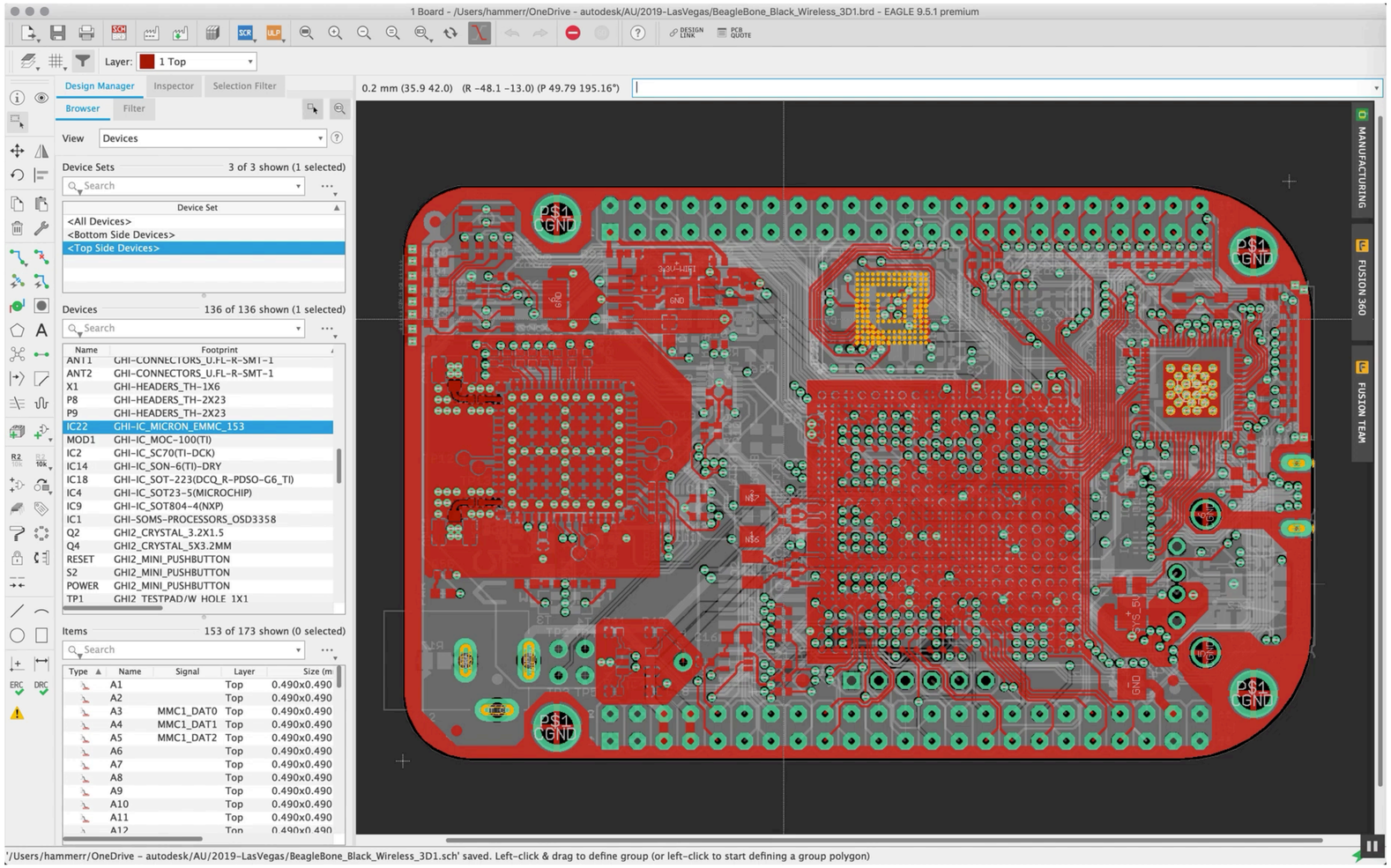
Use PCB Object from Fusion 360



A PCB is derived from the enclosure in Fusion 360 and used in EAGLE.

https://www.screencast.com/t/2ptlQZOhPcpE

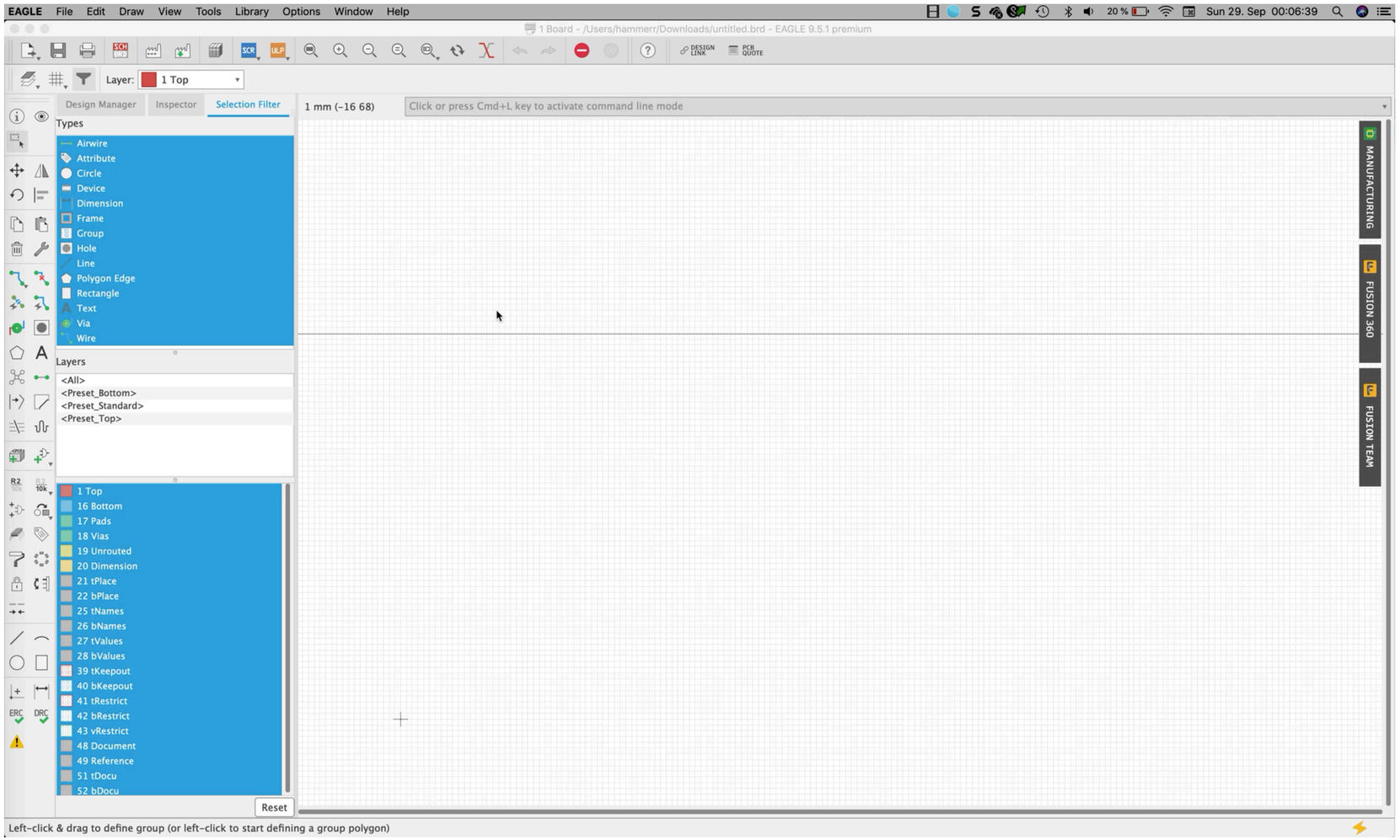
EAGLE to Fusion Collaboration



Use EAGLE layout in Fusion 360

https://www.screencast.com/t/zTWt3PN2XlJv

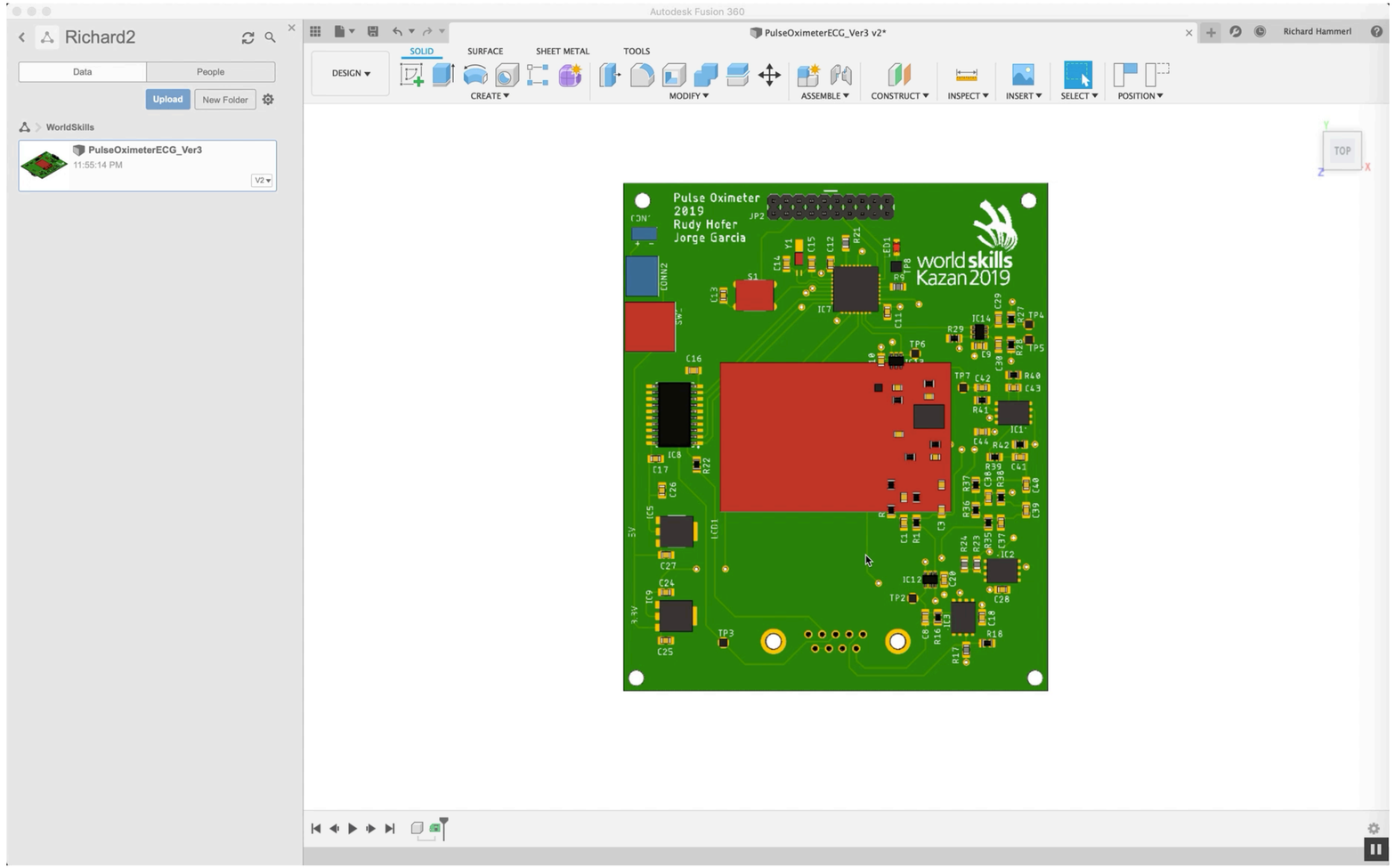
DXF Import



Import from an external DXF file

https://www.screencast.com/t/p3t011XFtI

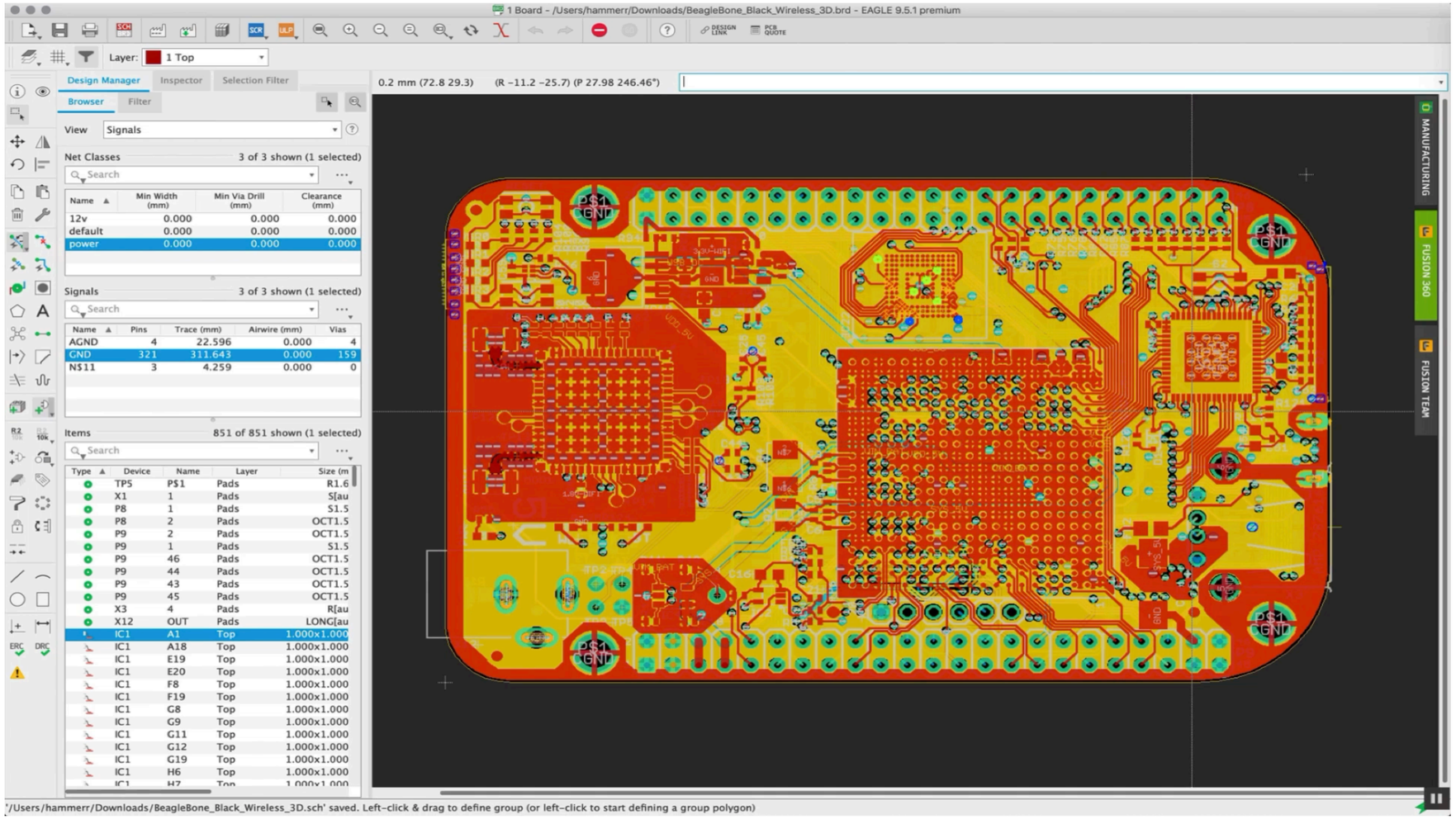
PCB Design Changes



Move components in Fusion and "Pull" in EAGLE

https://www.screencast.com/t/R8rrAphuJbhH

Fusion Team



Share projects and data with others

https://www.screencast.com/t/Sh8oLZpVK

Closing…

EAGLE related classes at AU 2019

CP321596 - Creating a Pulse Oximeter: Utilizing the Cloud to Prototype in EAGLE and Fusion 360

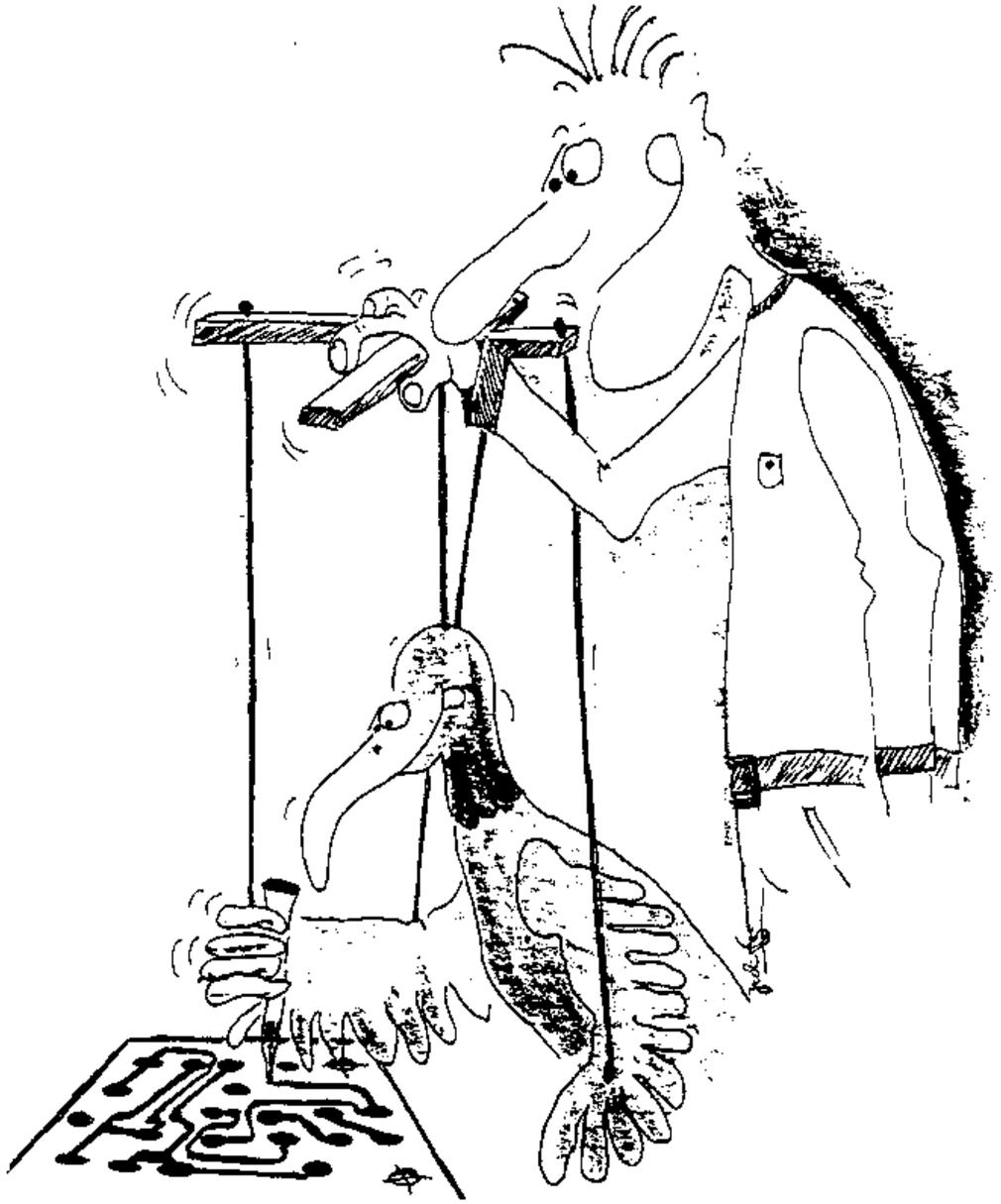
Wednesday, Nov 20, 10:30 AM - 11:30 AM – Lido 3005, Level 3 -- 60 MINUTES INSTRUCTIONAL DEMO, James Youmatz, Edwin Robledo

CES323493 - Global Engineering Design Tools

Wednesday, Nov 20, 9:15 AM - 10:15 AM – Marcello 4405, Level 4 -- 60 MINUTES INDUSTRY TALK, Taylor Sharpe

And some more about Fusion 360. Look for Fusion in your AU app.

Thank you for your attention!



AUTODESK®

Make anything™