Using Models, Simulation and Measurements for Teaching Circuit Design

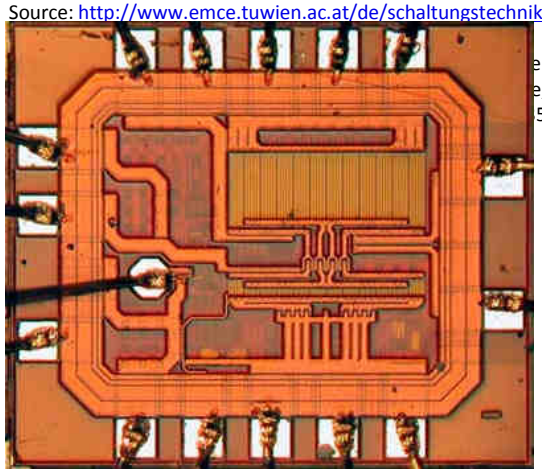
Prof. Dr.-Ing. Jörg Vollrath

Outline

- Introduction to Teaching Circuit Design
  - Objective
  - Class Content
- Tools and Application
  - LTSPICE, Electronic Explorer
  - Class and Laboratory
  - Amplifier Example
- Evaluation and Outlook

Teaching Circuit Design

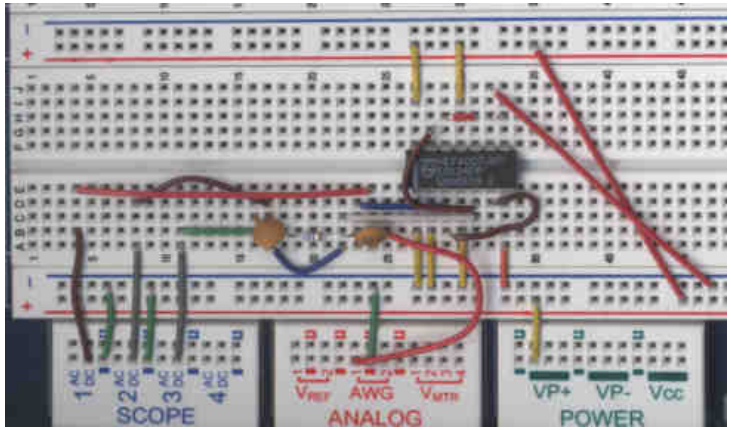
Source: http://www.emce.tuwien.ac.at/de/schaltungstechnik.htm



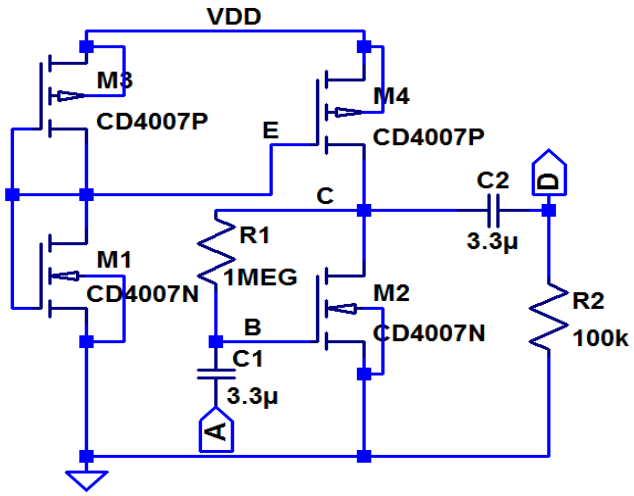
Integrated Circuit

Transfer function:

$$\underline{V}_{u3} = \frac{\underline{u}_a}{\underline{u}_e} = -\frac{R_C}{\frac{1}{g_m} + \frac{1}{\frac{1}{R_E} + j\omega C}}$$

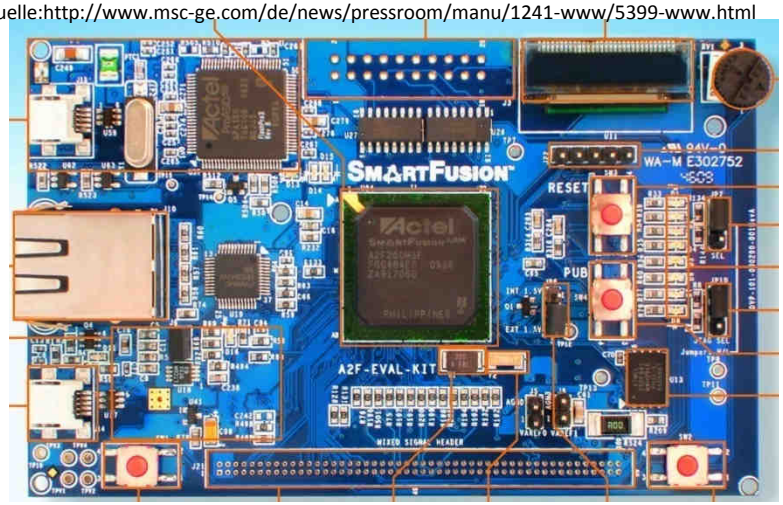


Testboard



Schematic

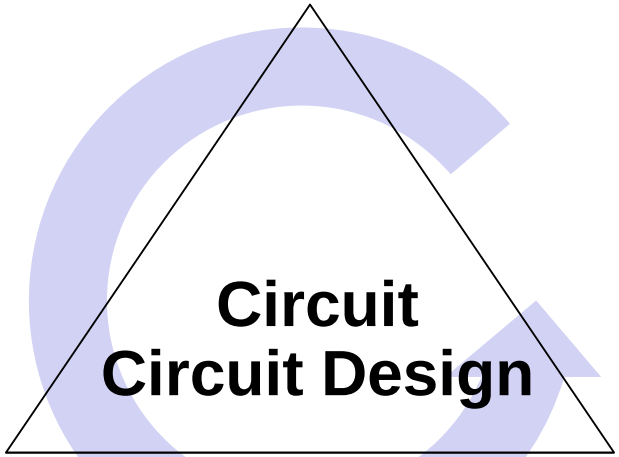
Quelle:http://www.msc-ge.com/de/news/pressroom/manu/1241-www/5399-www.html



Final PCB

Class Content: Circuit Design

Model
- Equation
- Equivalent Circuit
- Data sheet



Simulation
- SPICE
- Multisim

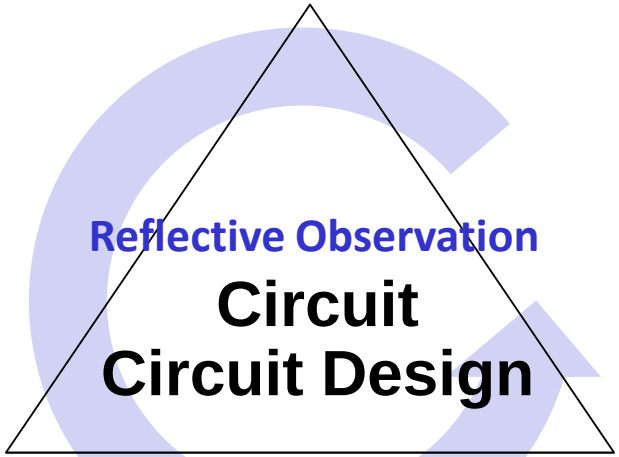
Measurement
- Verification
- Test
- Electrical Data

Class Content: Circuit Design

Abstract Thinking

Model

- Equation
- Equivalent Circuit
- Data sheet



Reflective Observation

Circuit
Circuit Design

Concrete Experience

Simulation

- SPICE
- Multisim

Active documentation

Concrete Experience

Measurement

- Verification
- Test
- Electrical Data

Active documentation

Kolb's Learning model

Challenge

- Circuit design uses models, simulation and measurement.
- Students have to learn to apply these tools properly.

- **How can we teach it?**

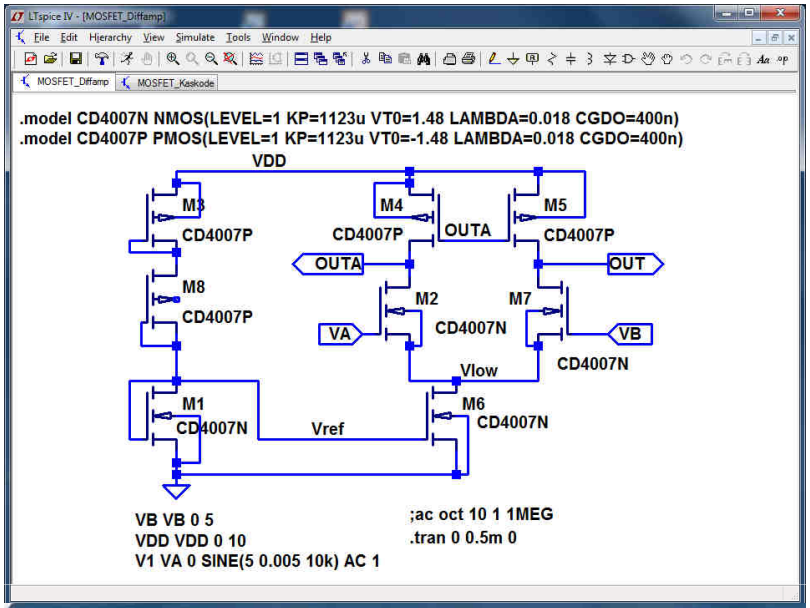
- Classic approach: lecture and high cost laboratory
- Web based approach: lecture and remote laboratory

- Use of „no cost“, portable tools during class, laboratory and at home.
  - Simulation: LTSPICE (Multisim)
  - Measurement: Electronic Explorer (MyDAQ)
  - Data Analysis: Excel (Matlab)

Simulation, Measurement and Theory

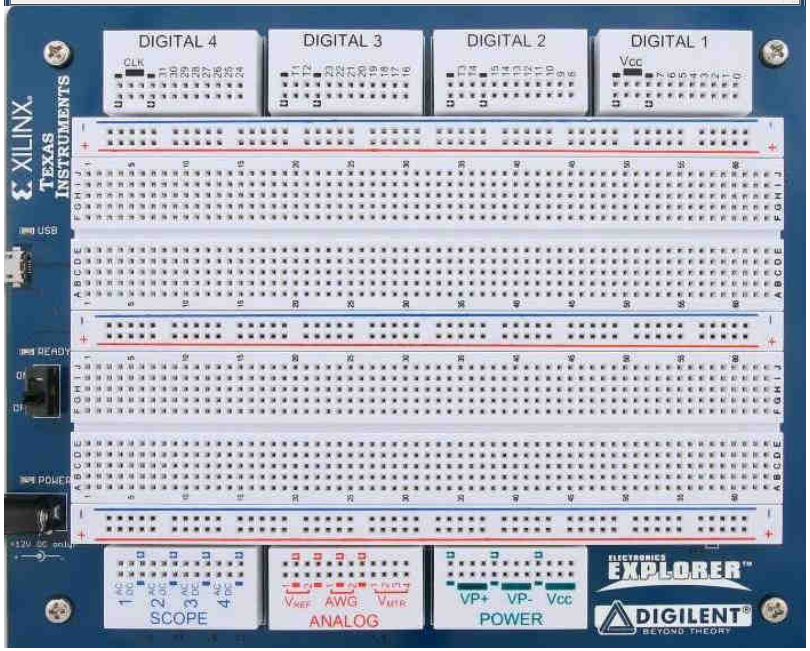
Circuit Simulator

- Schematic of circuit
- LTSPICE, PSPICE, Multisim



Measurement

- Digilent (Trenz) Electronic Explorer 350.-
- 4 channel Oscilloscope
- 32 Digital IO
- 2 Arbitrary waveform generator
- Power supply: 5V, +12V,-12V
- Breadboard
- Free software



Data Analysis and Model in Excel

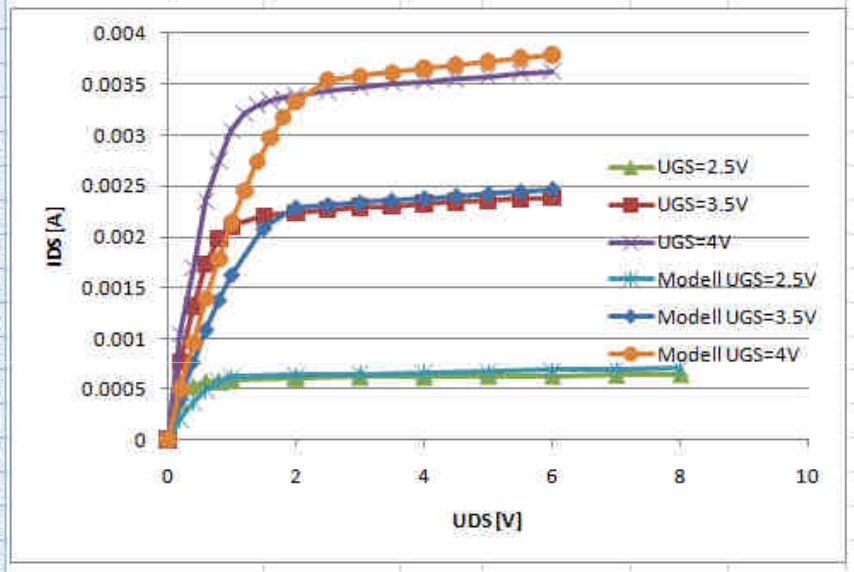
- MOSFET Transistor
  - Transistorequation
    - Parameter: V_{Th}, β, λ

$$I_{DS} = \beta (U_{GS} - V_{th})^2 (1 + \lambda U_{DS})$$

  - Measurements
    - I(U) Curve
  - Simulation
    - I(U) Curve

- Matching
  - Graph
  - Relative error

	A	B	C	D	E	F	G
1	MOSFET	4007					
2	V_{Th}	1.4					
3	β	0.0005					
4	λ	0.02					
5	Summe Fehlerquadrate	2.00E+00					



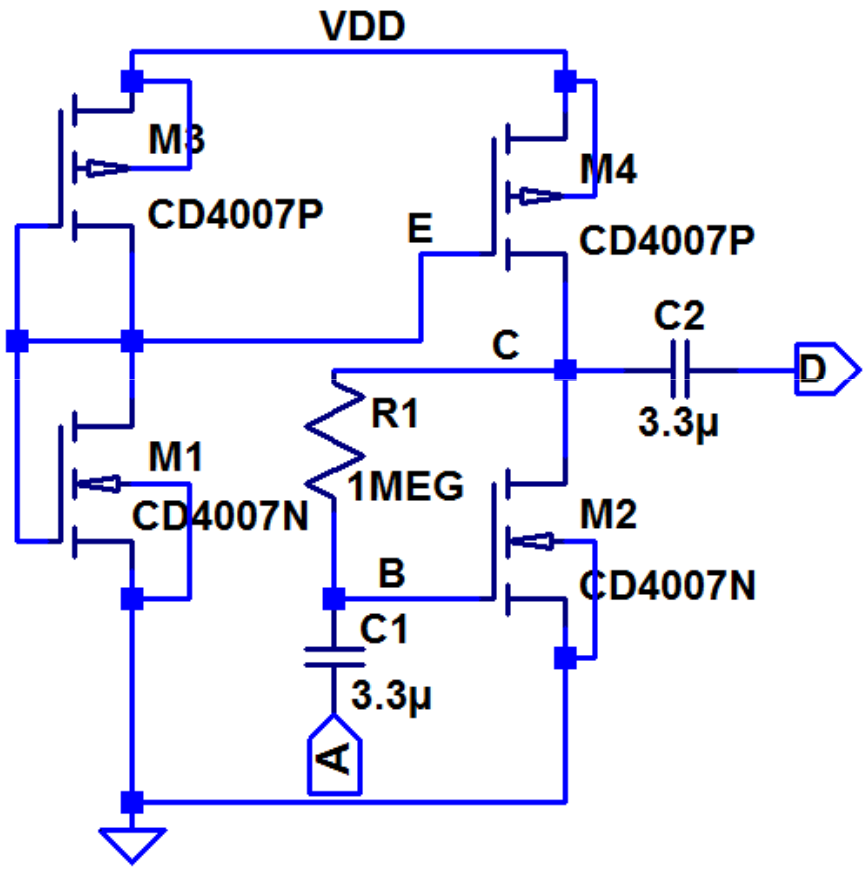
	UDS	UGS	Umess	Modell UGS=3	Rmess	UGS=3.5V	Fehlerquadrate
24	0	3.5	0	0	1103	0	
25	0.2	3.5	0.85	0.0004016	1103	0.0007706	8.44E-01
26	0.4	3.5	1.46	0.00076608	1103	0.0013237	5.30E-01
27	0.6	3.5	1.91	0.00109296	1103	0.0017316	3.41E-01
28	0.8	3.5	2.19	0.00138176	1103	0.0019855	1.91E-01
29	1	3.5	2.32	0.001632	1103	0.0021034	8.34E-02
30	1.5	3.5	2.43	0.00208575	1103	0.0022031	3.16E-03
31	2	3.5	2.47	0.002288	1103	0.0022393	4.52E-04
32	2.5	3.5	2.5	0.00231525	1103	0.0022665	4.43E-04
33	3	3.5	2.53	0.0023373	1103	0.0022937	3.47E-04
34	3.49	3.5	2.54	0.002358909	1103	0.0023028	5.66E-04

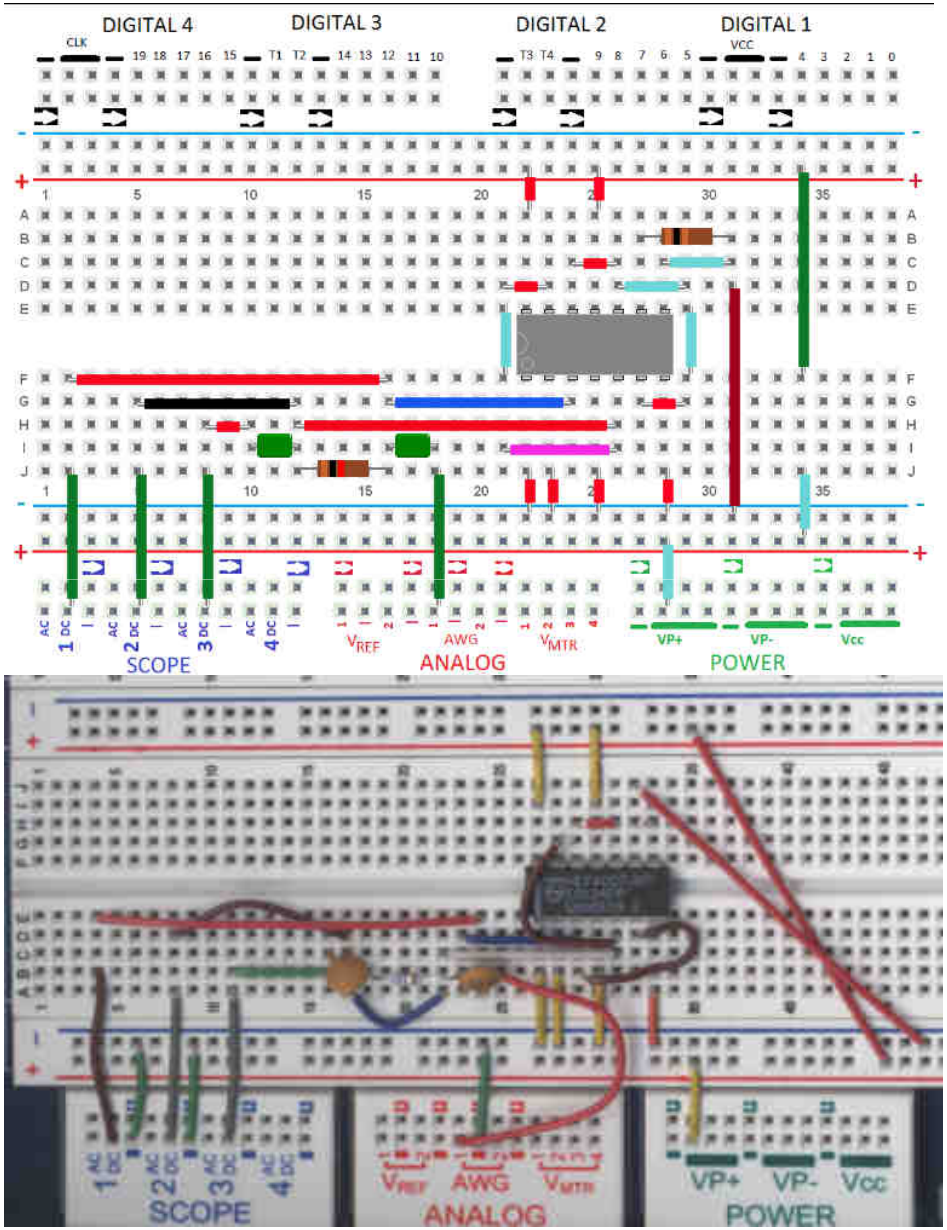
Laboratory (List)

- Feedback
  - Problems can be verified using SPICE and measurements
  - complex laboratories can be solved
- Methods
  - Repetition
  - Documenting results in an electronic book
- Competencies
  - Verification of results
  - Critical thinking

1. Data analysis using Excel
2. Diode measurement
3. Diode equation in SPICE and Excel
4. MOSFET Transistor measurement
5. MOSFET equations in SPICE and Excel
6. Bipolar Transistor
7. MOSFET as amplifier
8. Differential amplifier in SPICE
9. Operational Amplifier (OpAmp)
10. OpAmp in SPICE
11. Reporting

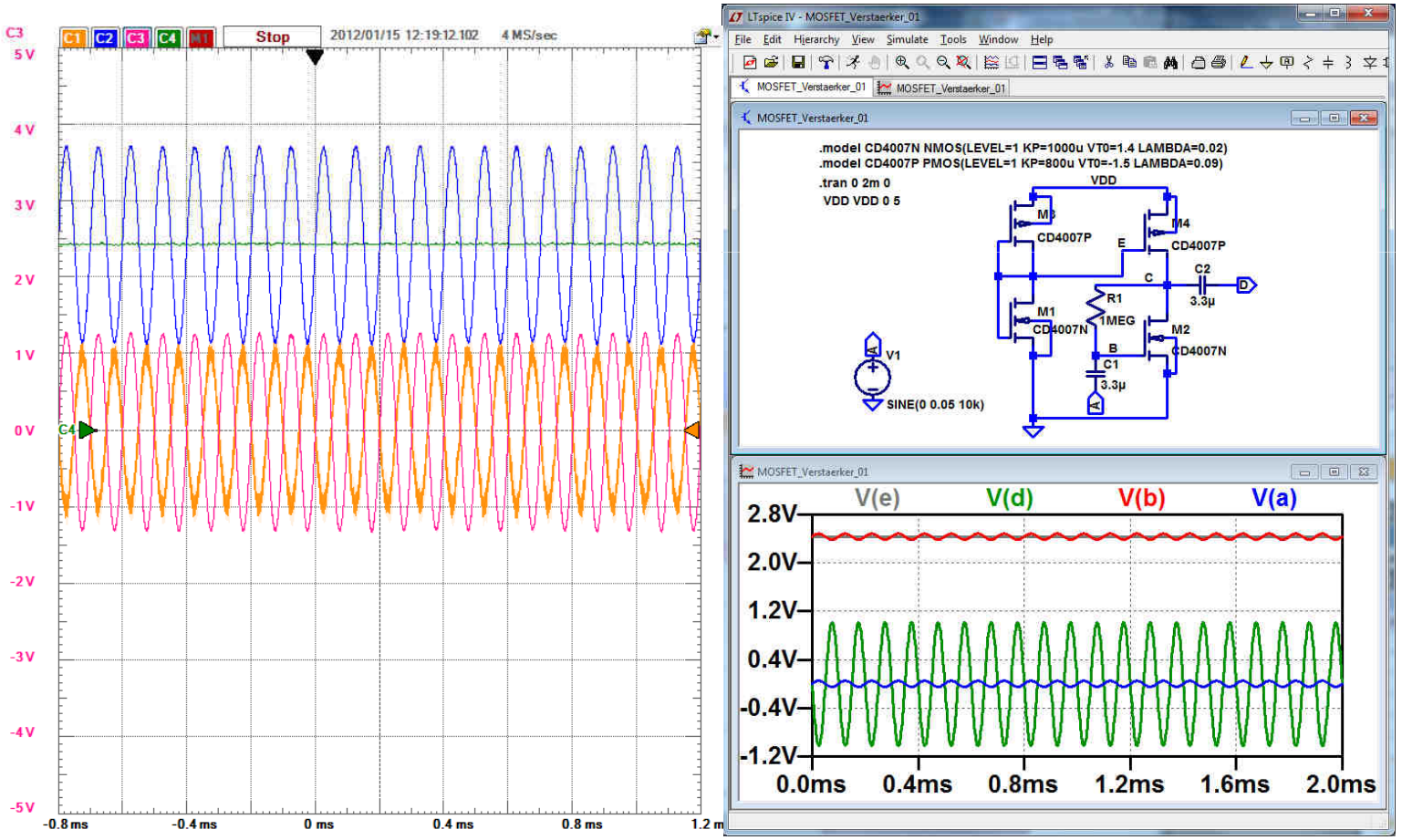
Laboratory MOSFET Amplifier





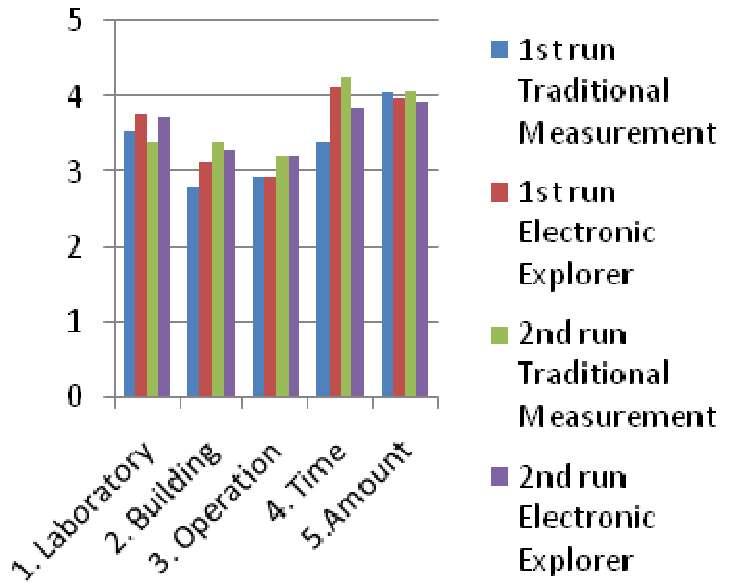
Comparison of Results: Calculation (Theory), Measurement,SPICE Simulation

	Calculation	Measurement	SPICE
V_E (V)	2.39	2.44	2.43
A_V	-18	-22	-20.4

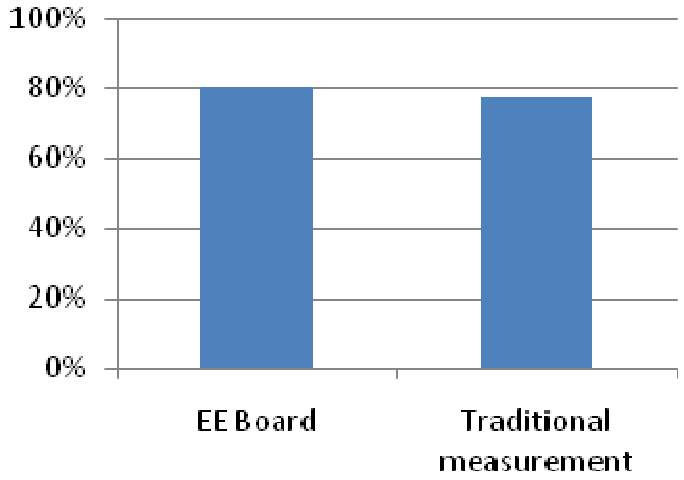


Evaluation

- Number of students:46
- 3 Groups
- Scale 1..5
- Laboratory exercise was
  - Difficult 5, 4, 3, 2, 1 Easy
- Building the circuit was
  - Difficult 5, 4, 3, 2, 1 Easy
- Operation of measurement equipment was
  - Difficult 5, 4, 3, 2, 1 Easy
- How much time did you need for measurements?
  - A lot 5, 4, 3, 2, 1 Little
- How do you rate the amount of work given the laboratory time?
  - A lot 5, 4, 3, 2, 1 Little
- Quality of measurement results



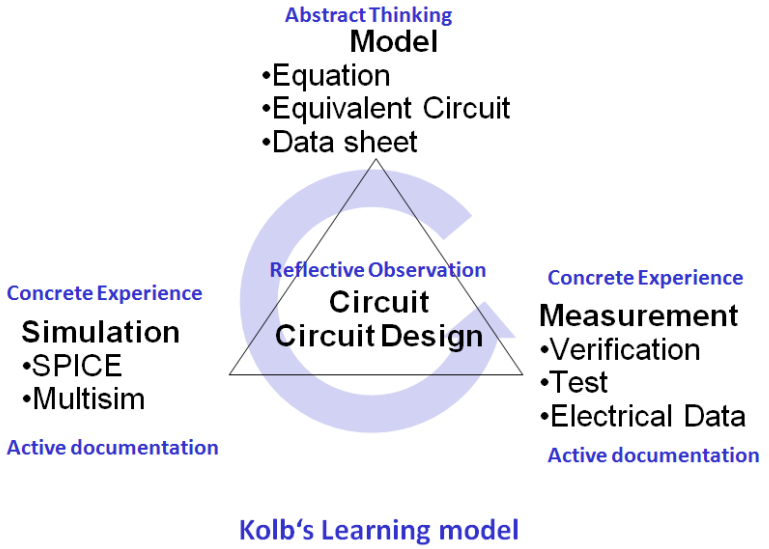
Good quality of measurements



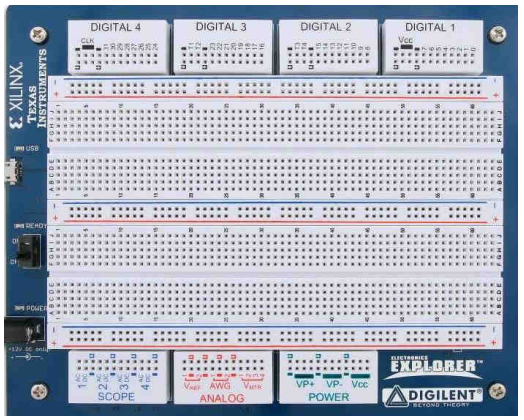
- Low number of students
- Different instructions

Summary and Outlook

- Circuit Design
  - **Model, Simulation, Measurement**
- Low cost environment
  - LTSPICE, Electronic Explorer
  - Students have more access and motivation
  - Repetitive use
- Competencies for problem solving
  - Measurement based modeling
  - Circuit ideas can be simulated
  - Measurement and verification of ideas
  - **Reflective Thinking**
- More investigations are needed to improve the set up.
- **Low cost equipment improves classes and laboratories for deep learning**



Kolb's Learning model



Thank You!