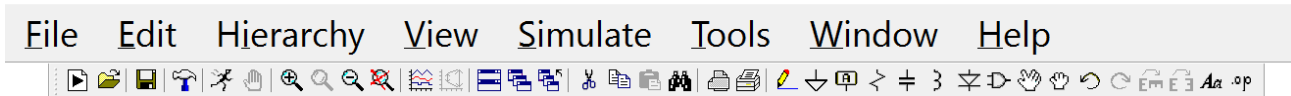Faculty of Engineering and Mathematics

Quick Start Guide LTspice for EPU-Workshop 2021

Menu with underlying toolbar

LTspice XVII - Potentiometer-Schaltung.asc

File   Edit   Hierarchy   View   Simulate   Tools   Window   Help

Procedure:

Before the workshop:

- Download and install LTspice for free
    - https://www.analog.com/en/design-center/design-tools-and-calculators/ltspice-simulator.htm
- Create a new folder (for example. \ET_EPU_Workshop)
- The circuit to be created (*.asc) is also stored in this folder

During the workshop:

- Start LTspice
    - File > New schematic or <Ctrl + N>
    - File save as ... > MyFile.asc (to the ET_EPU_Workshop folder)
- Drawing circuit
    - Electronic components
        - Edit > Component or <F2>
        - For example enter European Resistor -> select, OK and position in circuit (mouse click), exit with <Esc>.
        - As a choice is also possible to use English Resistor
    - Add more components and connecting lines
        - Edit > Draw Wire or <F3> for connecting wires
        - Edit > Select Component or Parts from Toolbar
    - Important: Also add reference point (Ground, GND)
- Assign values to the components with the right mouse button
    - Voltage source DC value[V] 5 for 5 V, Series Resistance[Ω] 0, if ideal
        - Advanced for AC voltage (sine, pulse, ...), (none) for DC voltage, DC
    - Resistor 0.22k or 220 for 220 Ω, etc.)
    - Simulation of the circuit with Simulate > Run afterwards input transient analysis in Stop Time for example. 0.1 s
- After Simulate > Run graphic window appears
    - Signal curves are only visible when the measuring tip (probe) is clicked on the lines or the current clamp is positioned.

- Measuring tip for voltage signal, potential difference to GND
- Current clamp for current signal (position via component by mouse click)

  o The following files are created in the ET_EPU_Workshop folder (to be viewed with editor)
    - *.raw for drawing
    - *.log for log file
    - *.net for list with parts, parameters, instructions

Additional information:

- Set signal points
  - o Edit > Label Net or <F4> Net Name
  - o Enter number or text (signal name)
  - o Position on line
  - o Select reference point "ground" = GND (global note 0), has number 0
- Measurement of a voltage difference (not related to GND, e.g. with Wheatstone bridge)
  - o Set reference value (Mark Reference)
  - o View > Set Probe Reference or right mouse button: Mark Reference on signal line
    - First, however, simulation must be started with Simulate > Run
  - o Reset reference value (Mark Reference) when set to GND again
- Simulate > Edit Simulation Command
  - o Transient: Simulation in time domain