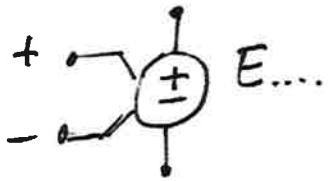
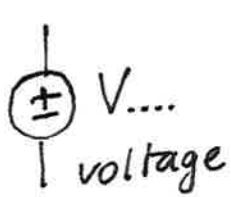
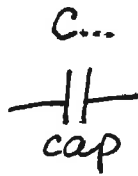
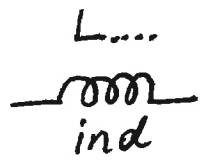
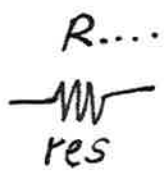
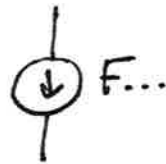


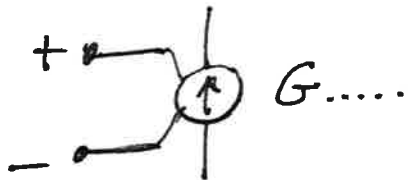
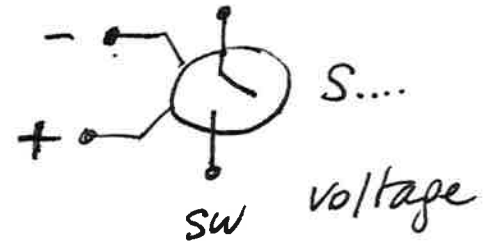
Getting started with SPICE



voltage dependent
voltage source
VCVS



current dependent
current source
CCCS

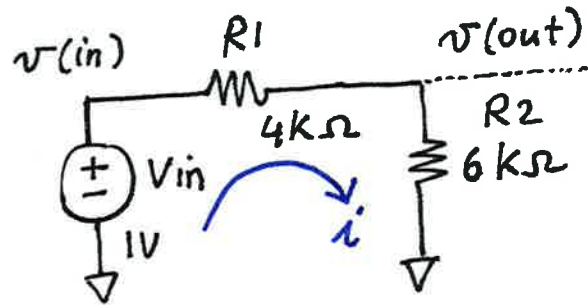


voltage dependent
current source
VCCS



current dependent
voltage source
CCVS

example



$$V_{out} = \frac{V_{in}}{R_1 + R_2} \cdot R_2 = 0.6V$$

$$i = \frac{V_{in}}{R_1 + R_2} = 100\mu A$$

LTSPICE shortcuts

CTRL-R	rotate component
CTRL-E	mirror component
F2	add component
F3	add wire
F4	add label
F5	delete
F6	duplicate
F7	move
F8	drag
F9	undo
↑F9	redo

G add ground
S add spice directive
↓
right click for help
on writing the
directive

T add a comment

TYPES OF SOURCES

- DC
- AC
- SINE
- PULSE
- EXP
- PWL

TYPES OF ANALYSIS

- OP
- TRAN
- AC
- TF
- DC

Right click → View → Spice Netlist
Spice Error Log

T → to change preferences

⚡ → to run simulation

↳ right click → add trace
(in the waveform window)
to plot the desired signals