

LTspice

Software program for kredsløbssimulering
Kan frit downloades fra
<http://www.linear.com/download>

v/ OZ5BG – Bent Grønbæk Olesen

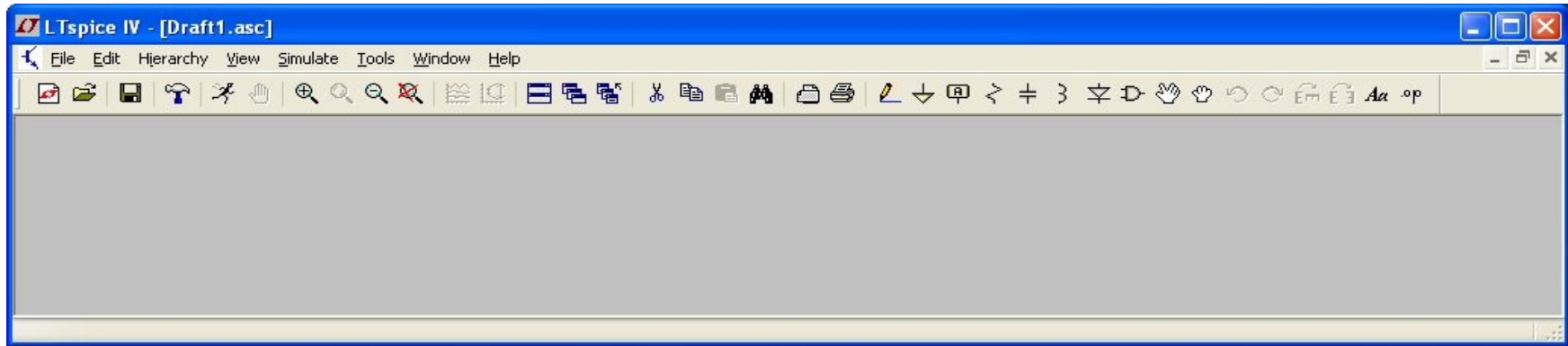


LTspice



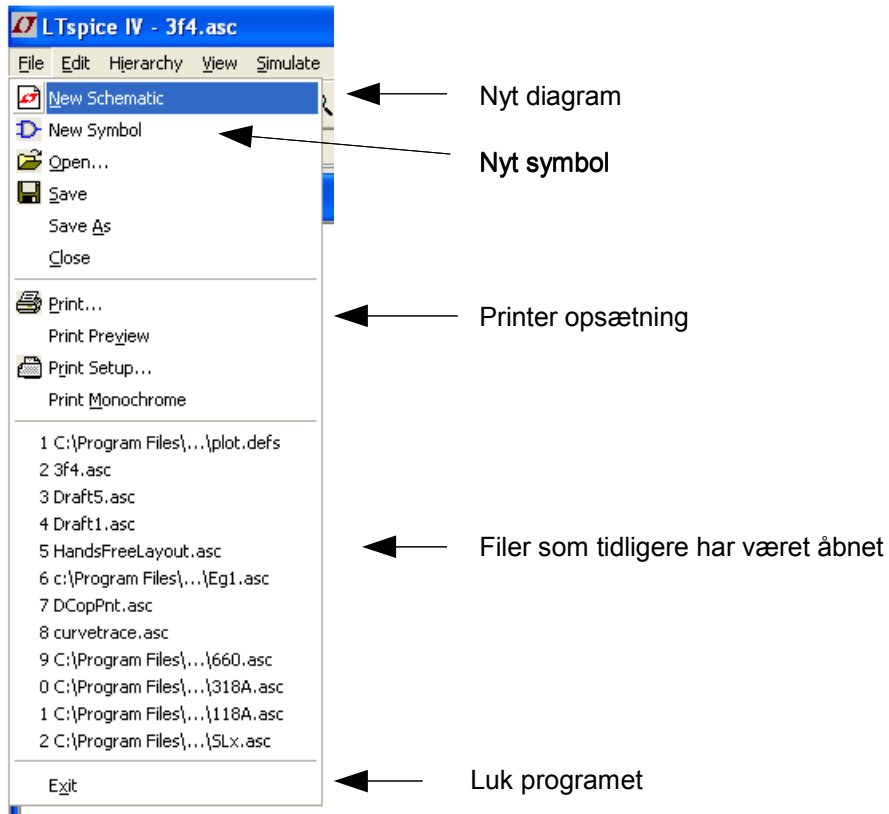
- Programmet er udviklet af Mike Engelhard
- Ltspice anvendes internt af Linear Technology.
- Der er modeller til de fleste af LT's IC'er.
- Kan anvendes som generelt spice simulator, ikke kun til Linear Technology IC'er.
- Ltspice koden er optimeret til nutidens multi-core processorer – derfor hurtig.

Diagram menuer

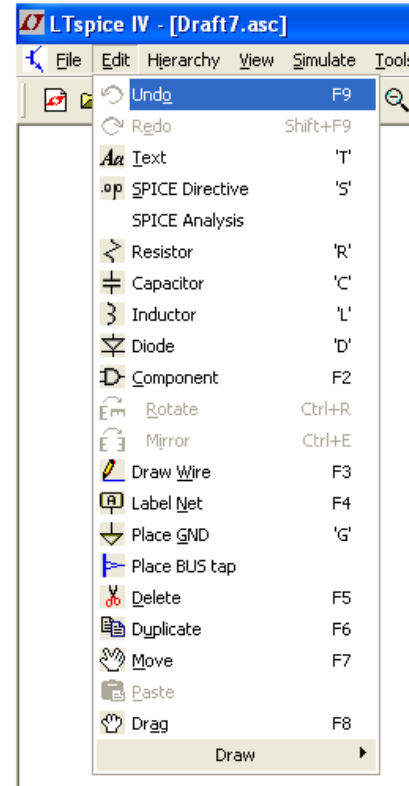


Schematic/Diagram menuer

File menu



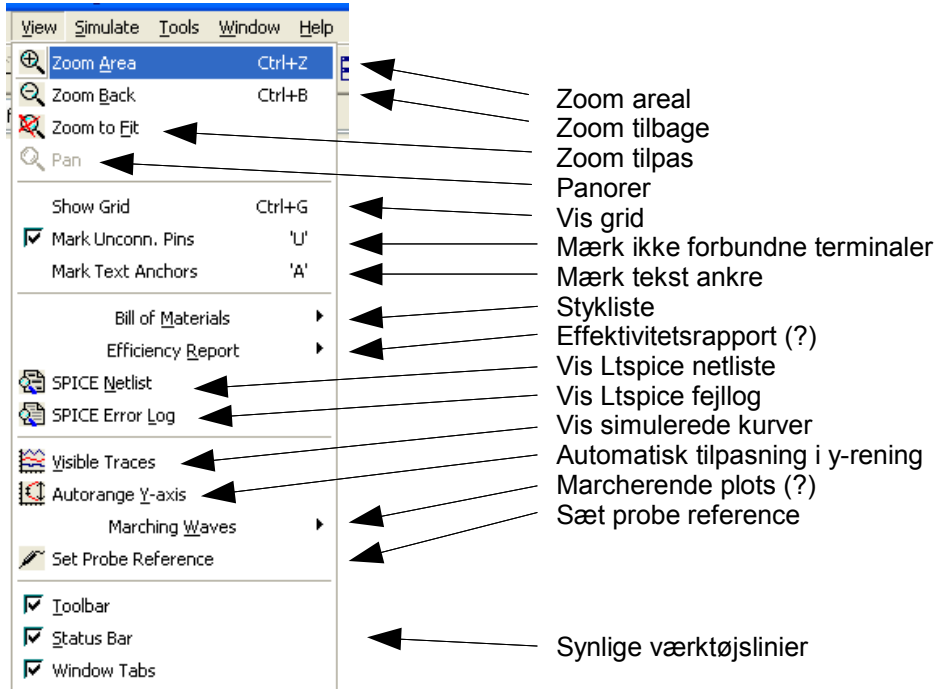
Rediger menu



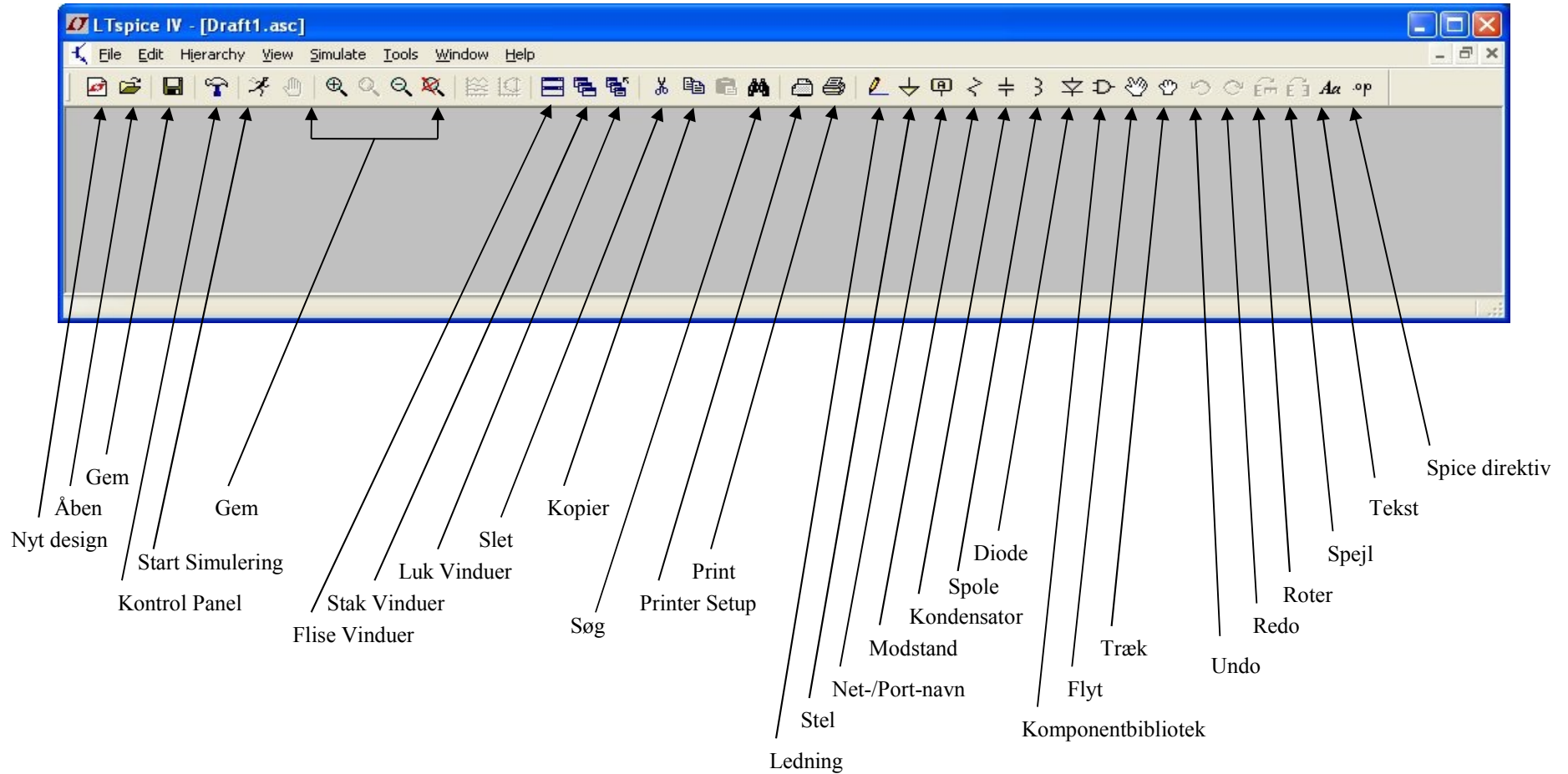
- Fortryd
- Gengør (Fortryd fortryd)
- Tekst
- Spice direktiv
- Spice analyse DC/AC/TRAN
- Modstand
- Kondensator
- Spole
- Diode
- Komponent (fra bibliotek)
- Roter
- Spejlvend
- Tegn Ledning
- Netnavn
- Placer stelsymbol
- Placer BUS tilledning
- Slet
- Kopier
- Flyt
- Indsæt
- Træk (inc. ledninger)

Schematic/Diagram menuer

View menu

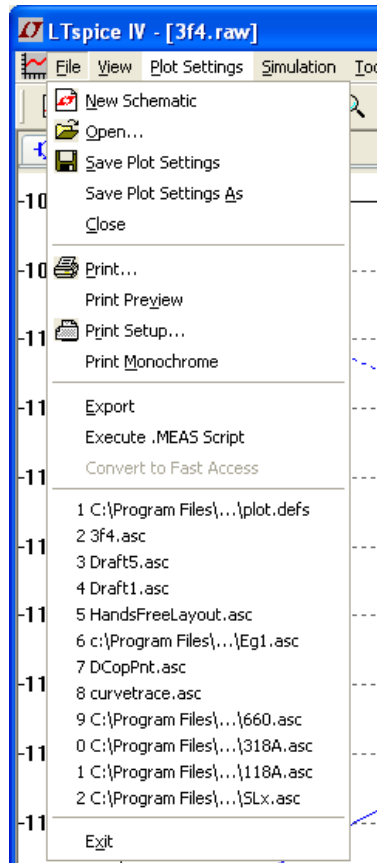


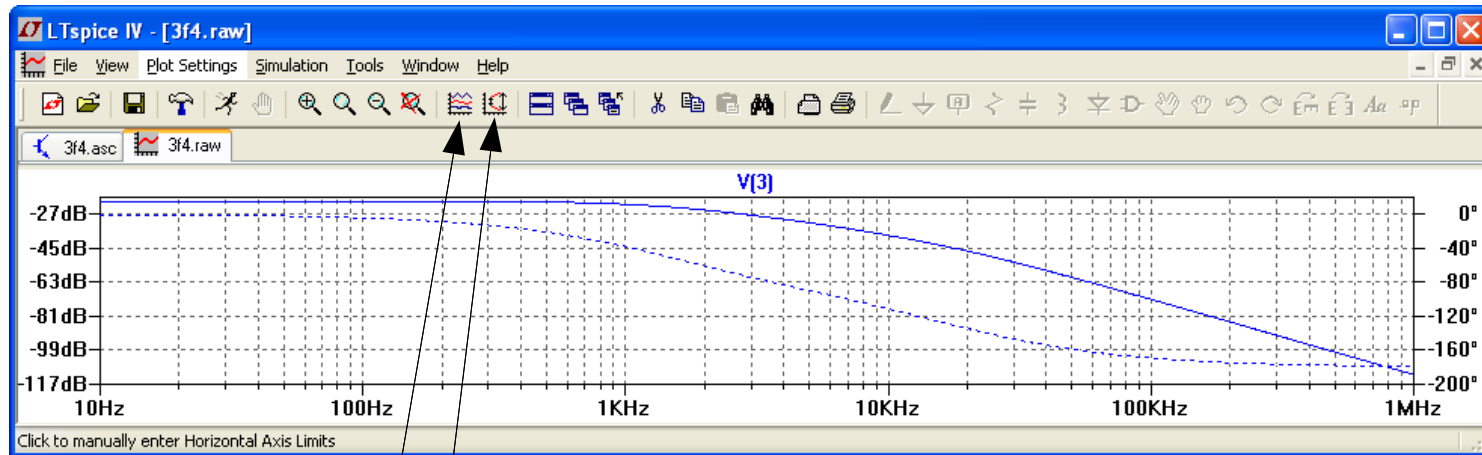
Ltspice menu



Plot menuer

File menu





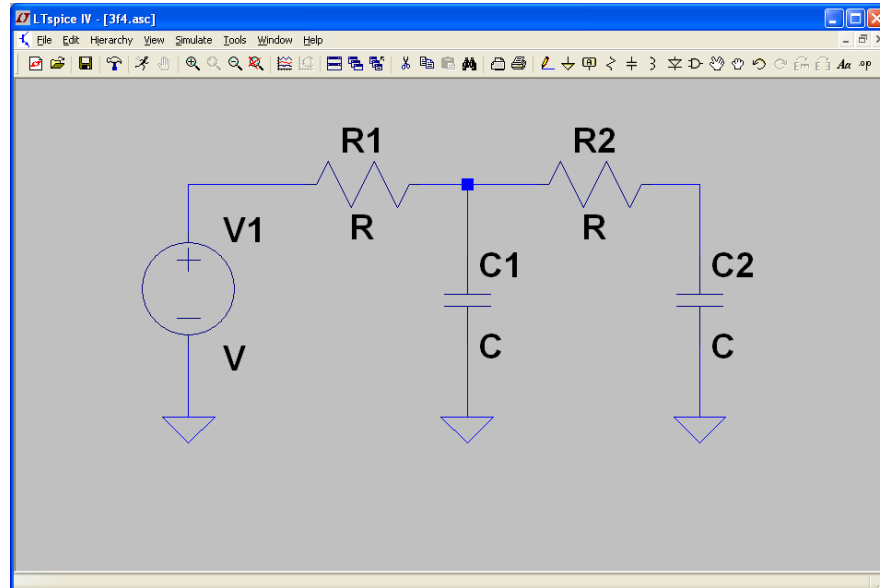
Tilpas i y-retning

Vælg kurve for plot

Vøres første kredsløb

- Placer modstande
- Placer kondensatorer
- Placer spændingsgenerator
- Placer stelsymboler

Der skal være mindst ét
Stelsymbol i et spice kredsløb



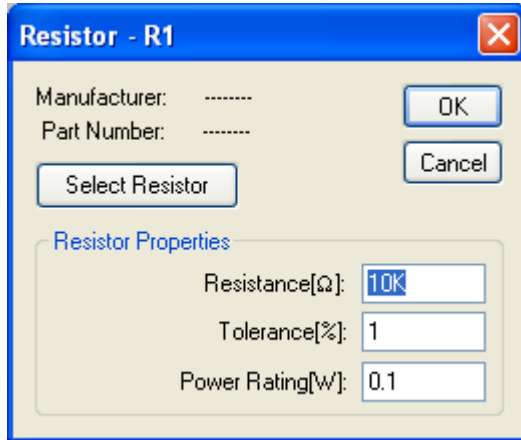
Der skal tilføjes værdier til modstande, kondensatorer og spændingsgenerator
Før kredsløbet er klar til simulering.

Tilret modstandsværdi

Metode 1

Hold cursor hen over modstand indtil der ommer en hånd,
Højre-klik for at få denne menu frem. Indfør rettelser og tryk OK

Klik på "Select Resistor" for at få en liste med modstandsværdier



Resistor - R1

Manufacturer: OK

Part Number: Cancel

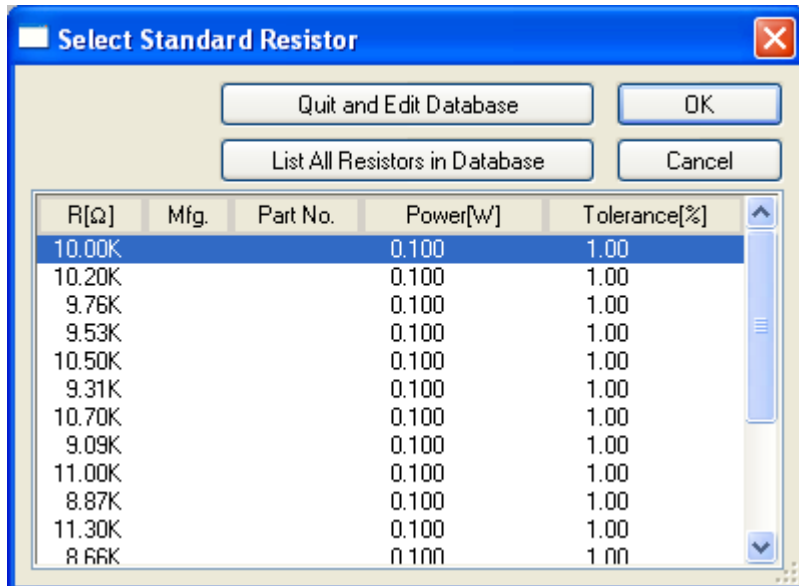
Select Resistor

Resistor Properties

Resistance[Ω]: 10K

Tolerance[%]: 1

Power Rating[W]: 0.1



Select Standard Resistor

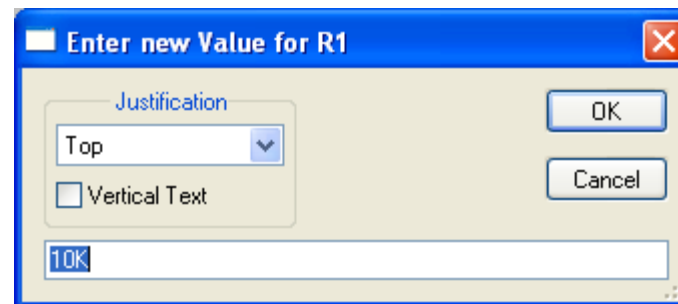
Quit and Edit Database OK

List All Resistors in Database Cancel

R[Ω]	Mfg.	Part No.	Power[W]	Tolerance[%]
10.00K			0.100	1.00
10.20K			0.100	1.00
9.76K			0.100	1.00
9.53K			0.100	1.00
10.50K			0.100	1.00
9.31K			0.100	1.00
10.70K			0.100	1.00
9.09K			0.100	1.00
11.00K			0.100	1.00
8.87K			0.100	1.00
11.30K			0.100	1.00
8.66K			0.100	1.00

Metode 2

Flyt cursor ned over modstandsværdien "R", højre-klik
Og nedenstående vindue kommer frem. Ret værdi
Og tryk OK



Enter new Value for R1

Justification

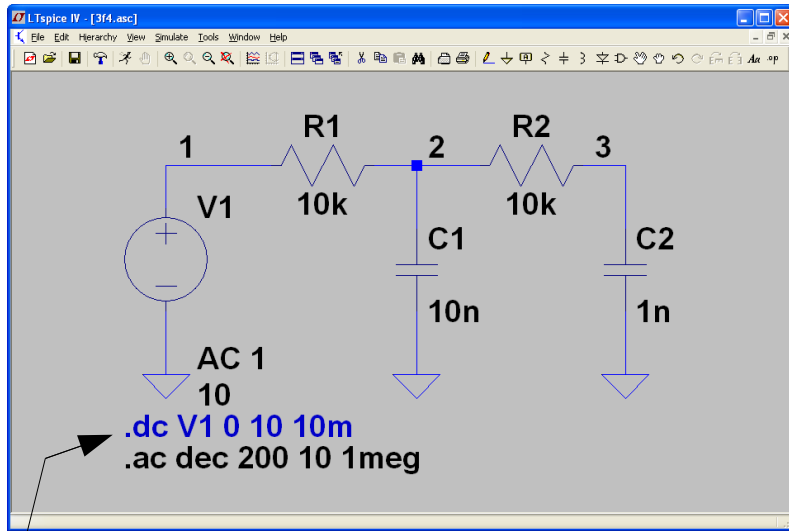
Top

Vertical Text

10K

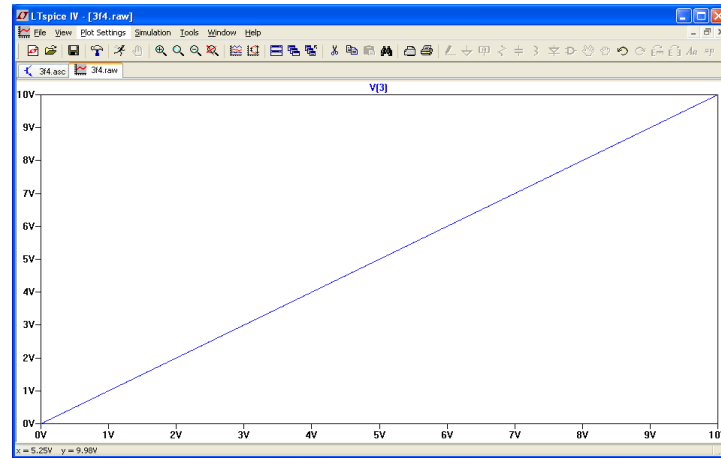
OK

Cancel

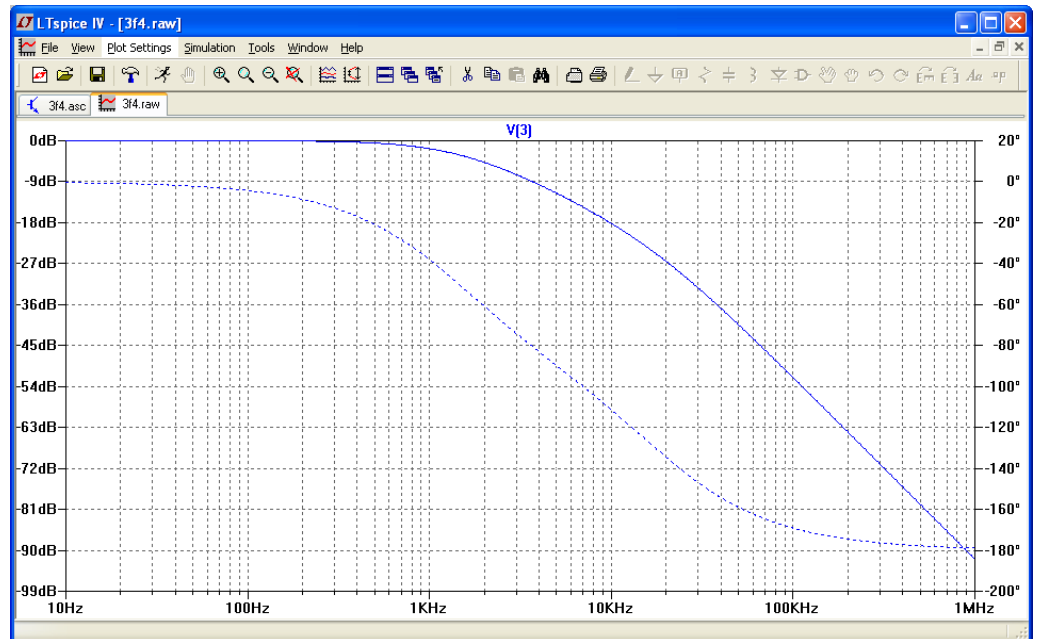


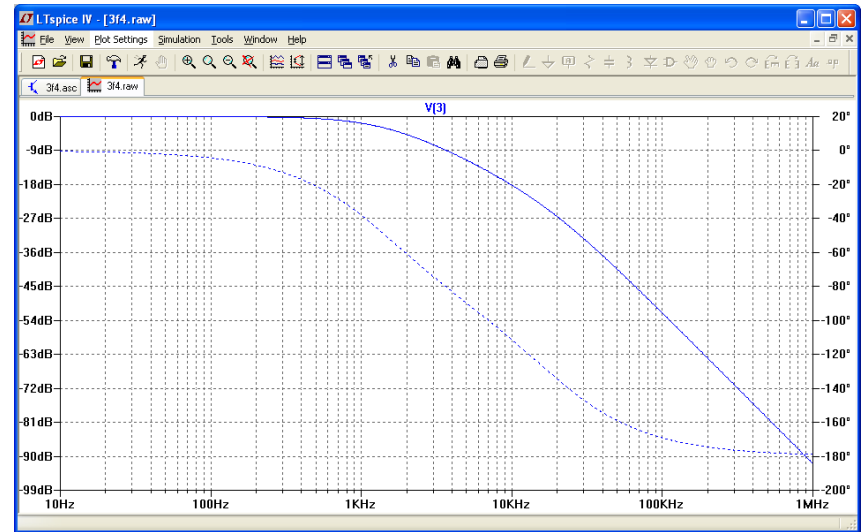
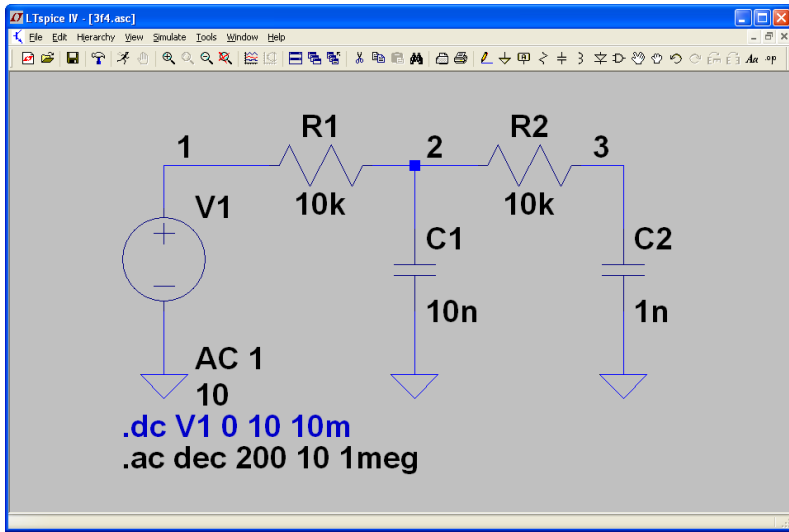
Højreklik på spice direktiv for at Ændre den til en kommentar og omvendt.

dB-værdier er i forhold til 1Vac

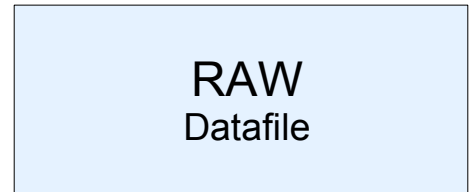
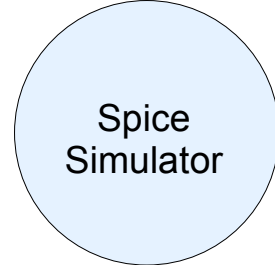
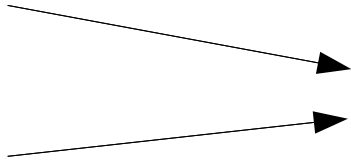


DC udgangsspænding er lig indgangsspænding da der ikke er nogen modstand til stel, dvs. der går ingen DC strøm.

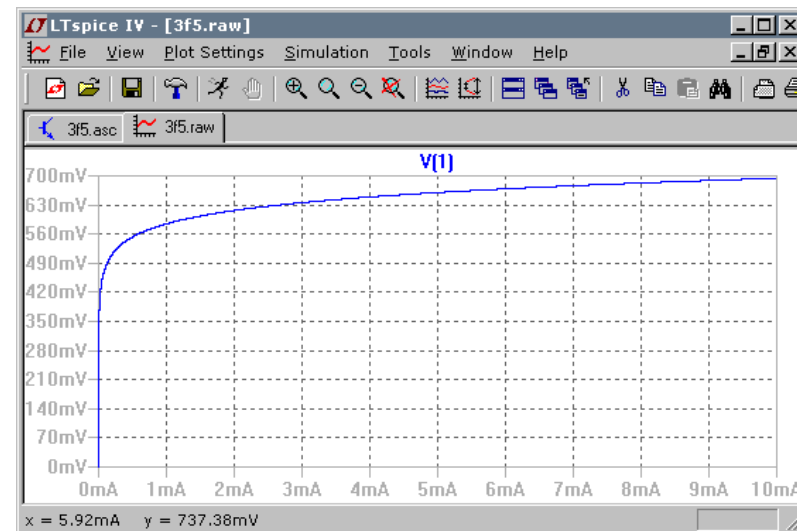
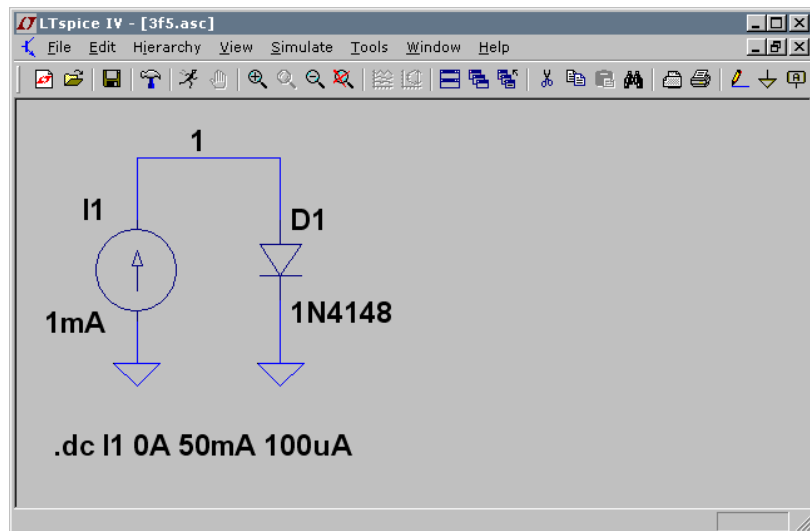




```
* D:\backup\pc-bgo3\3edr\Foredrag\Misc\doc00003\ltspice\3f4.asc
R1 2 1 10K tol=1 pwr=0.1
R2 3 2 10k
C2 3 0 1n
C1 2 0 10n
V1 1 0 10V AC 0.1V
* .dc V1 0V 10V 10mV
.ac dec 200 10 1meg
.backanno
.end
```



Spice model

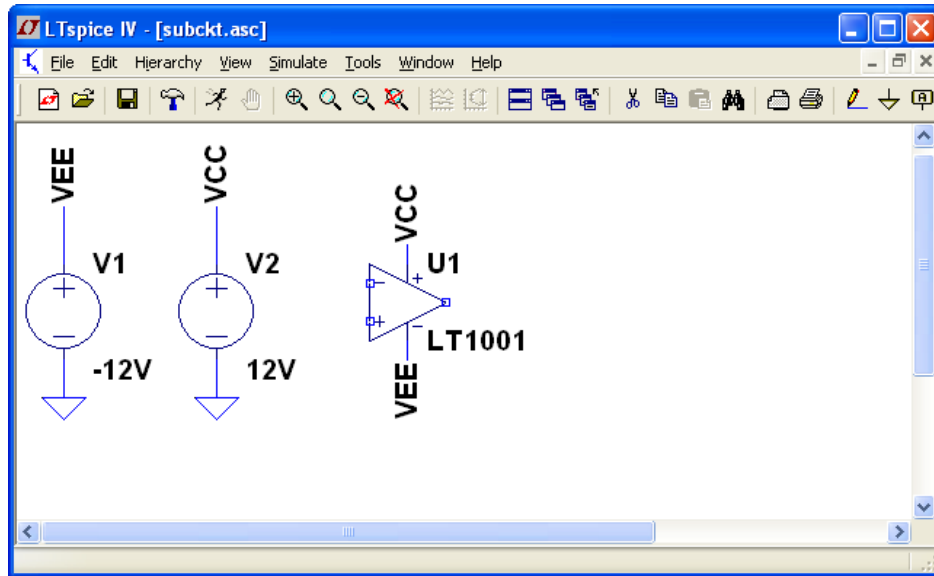


```
* W:\3edr\Foredrag\Misc\doc00003\ltspice\3f5.asc
I1 0 1 1mA
D1 1 0 1N4148
.model D D
.lib C:\PROGRA~1\LTC\LTSPIC~1\lib\cmp\standard.dio
.dc I1 0A 10mA 10nA
.backanno
.end
```

```
.model 1N4148 D(Is=2.52n Rs=.568 N=1.752 Cjo=4p M=.4 tt=20n Iave=200m Vpk=75 mfg=Motorola type=silicon)
```

Modelparametrene beskriver dioden 1N4148. D'et foran parantesen fortæller Ltspice at der er tale om en diodemodel.

Komponent med sub circuit



Findes i LTC.lib

```
.SUBCKT LT1001 3 2 7 4 6
* INPUT
RC1 7 80 6631
RC2 7 90 6631
Q1 80 102 10 QM1
Q2 90 103 11 QM2
RB1 2 102 500
RB2 3 103 500
DDM1 102 104 DM2
DDM3 104 103 DM2
DDM2 103 105 DM2
DDM4 105 102 DM2
C1 80 90 8.66e-12
RE1 10 12 1409
RE2 11 12 1409
IEE 12 4 9.901e-6
RE 12 0 20200000
CE 12 0 1.579E-12
* INTERMEDIATE
GCM 0 8 12 0 7.558E-11
GA 8 0 80 90 1.508E-04
R2 8 0 100000
C2 1 8 3e-11
GB 1 0 8 0 1538
* OUTPUT
RO1 1 6 25.75
RO2 1 0 34.25
RC 17 0 4.228e-6
GC 0 17 6 0 236500
D1 1 17 DM1
D2 17 1 DM1
D3 6 13 DM2
D4 14 6 DM2
VC 7 13 1.803
VE 14 4 1.803
IP 7 4 0.00159
DSUB 4 7 DM2
* MODELS
.MODEL QM1 NPN(IS=8e-16 BF=5500)
.MODEL QM2 NPN(IS=8.006E-16 BF=9900)
.MODEL DM1 D(IS=2.331e-8)
.MODEL DM2 D(IS=8e-16)
.ENDS LT1001
```

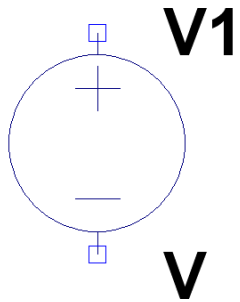
Spice model

Der findes modeller for følgende komponenttyper:

- (Modstande)
- (Kondensatorer)
- (Spoler)
- Spændingsstyret switch (SW)
- Strømstyret switch (CSW)
- Dioder (D)
- Bipolar transistorer (NPN/PNP)
- Junction FET (NJF/PJF)
- Mosfet (NMOS,PMOS og VDMOS)
- MeshFET (NMF, PMF)
- Transmissionslinie med og uden tab (LTLINE,TLINE)

Hvis man mangler en SPICE model for en komponent, så finder man den på nettet!
Det er ikke noget man umiddelbart selv laver.

Generator typer



Independent Voltage Source - V1

Functions

- (none)
- PULSE(V1 V2 Tdelay Trise Tfall Ton Period Ncycles)
- SINE(Voffset Vamp Freq Td Theta Phi Ncycles)
- EXP(V1 V2 Td1 Tau1 Td2 Tau2)
- SFFM(Voff Vamp Fcar MDI Fsig)
- PWL(t1 v1 t2 v2...)
- PWL FILE:

Make this information visible on schematic:

DC Value

DC value:

Make this information visible on schematic:

Small signal AC analysis(.AC)

AC Amplitude:

AC Phase:

Make this information visible on schematic:

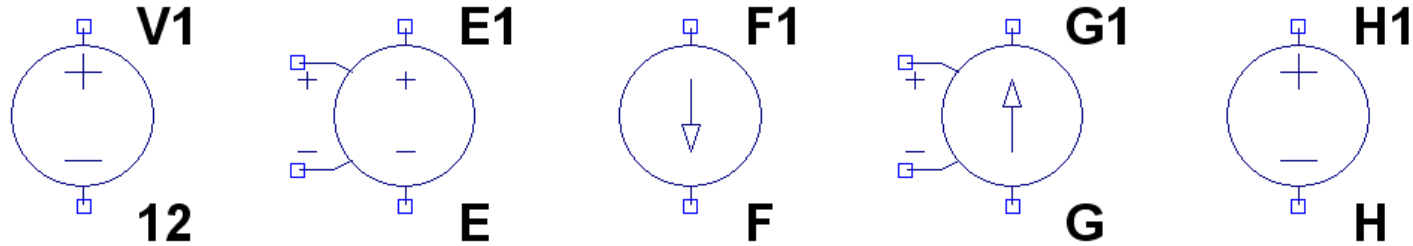
Parasitic Properties

Series Resistance[Ω]:

Parallel Capacitance[F]:

Make this information visible on schematic:

Generator typer



E: Spændingsstyret spændingsgenerator

Syntax: `Exxx n+ n- nc+ nc- <gain>`

F: Strømstyret strømgenerator

Syntax: `Fxxx n+ n- <Vnam> <gain>`

G: Spændingsstyret strømgenerator

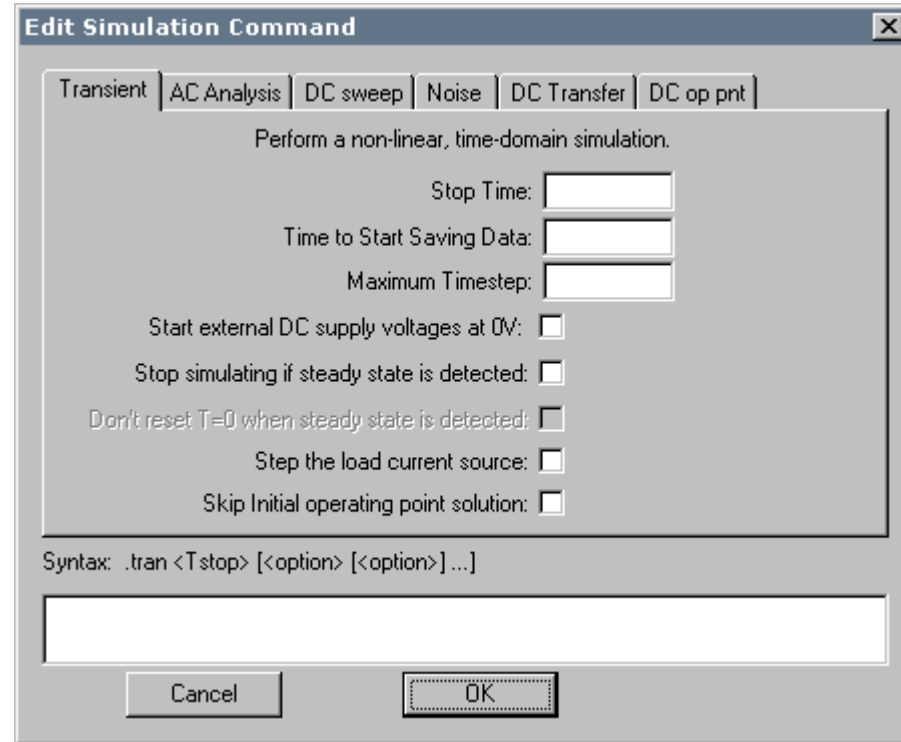
Syntax: `Gxxx n+ n- nc+ nc- <gain>`

H Strømstyret spændingsgenerator

Syntax: `Hxxx n+ n- <Vnam> <transresistance>`

Simulering modes:

1. DC simulering
2. AC simulering
3. Transient simulering



DC simulating

Edit Simulation Command [X]

Transient | AC Analysis | **DC sweep** | Noise | DC Transfer | DC op pnt

Compute the DC operating point of a circuit while stepping independent sources and treating capacitances as open circuits and inductances as short circuits.

1st Source | 2nd Source | 3rd Source

Name of 1st Source to Sweep:

Type of Sweep:

Start Value:

Stop Value:

Increment:

Syntax: `.dc <Source1> [<oct,dec,lin>] <Start> <Stop> [<Incr>][<source2> ...]`

Cancel OK

AC simulating

Edit Simulation Command [X]

Transient | **AC Analysis** | DC sweep | Noise | DC Transfer | DC op pnt

Compute the small signal AC behavior of the circuit linearized about its DC operating point.

Type of Sweep:

Number of points per decade:

Start Frequency:

Stop Frequency:

Syntax: `.ac <oct, dec, lin> <Npoints> <StartFreq> <EndFreq>`

Cancel OK

Transient simulating

Edit Simulation Command

Transient | AC Analysis | DC sweep | Noise | DC Transfer | DC op pnt

Perform a non-linear, time-domain simulation.

Stop Time: 100ms

Time to Start Saving Data: 0

Maximum Timestep: 1u

Start external DC supply voltages at 0V:

Stop simulating if steady state is detected:

Don't reset T=0 when steady state is detected:

Step the load current source:

Skip Initial operating point solution:

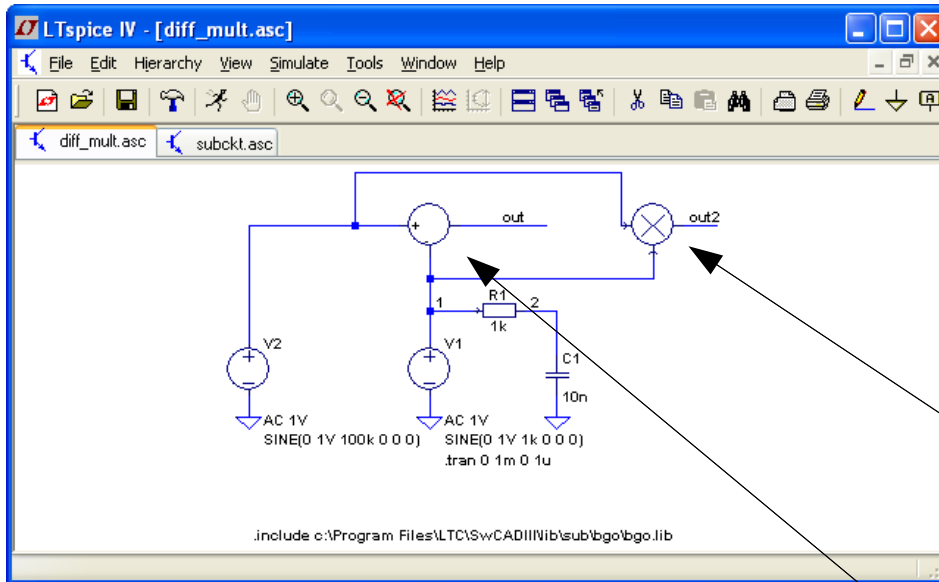
Syntax: .tran <Tprint> <Tstop> [<Tstart> [<Tmaxstep>]] [<option> [<option>] ...]

.tran 0 100ms 0 1u steady startup

Cancel OK

Egne symboler

Det er muligt at lave sine egne symboler for "dimser" som ikke findes i Ltspice



```
.subckt diff      1  2  3
r1 1 0 1g
r2 2 0 1g
b1 3 0 v=v(1)-v(2)
.ends
```

```
.subckt mult      1  2  3
r1 1 0 1g
r2 2 0 1g
b1 3 0 v=v(1)*v(2)
.ends
```

mult

diff