

Analogue Simulation

Module: SE3AE11 Analogue Simulation

Lecturer: James Grimbleby

URL: <http://www.personal.rdg.ac.uk/~stsgrimb/>

email: j.b.grimbleby@reading.ac.uk

Number of Lectures: 10

Number of Labs: 10

Recommended course text:

Muhammad H. Rashid

Introduction to PSpice Using OrCAD for Circuits and
Electronics 3rd Edition

Analogue Simulation Course Text

Muhammad H. Rashid

Introduction to PSpice Using
OrCAD for Circuits and
Electronics 3rd Edition

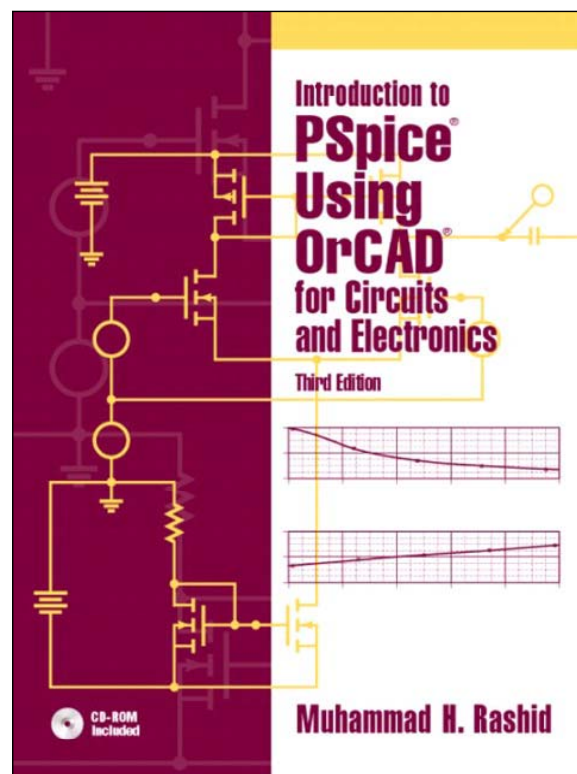
Prentice Hall 2003

ISBN: 0131019880

Includes:

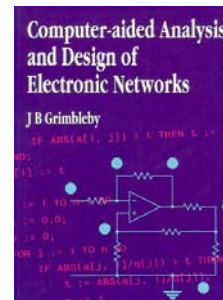
ORCAD Capture Lite 9.2

Price: £36 (approx)

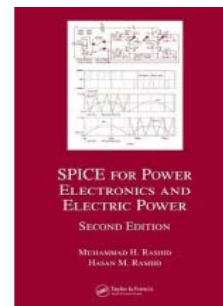


Analogue Simulation Further Reading

J. B. Grimbleby
Computer-Aided Analysis and
Design of Electronic Networks
Pitman 1990
ISBN: 0-273-03148-1



M.H. Rashid
Spice Simulations for Power
Electronics
Taylor & Francis Ltd 2005
ISBN: 0849334187



Analogue Simulation Resources

SPICE device model files for use in this course:

[http://www.personal.rdg.ac.uk/~stsgrimb/teaching/
spice_library.zip](http://www.personal.rdg.ac.uk/~stsgrimb/teaching/spice_library.zip)

Free Orcad/Pspice demo (select: Lite Capture & PSpice only):

<http://www.cadence.com/products/orcad/Pages/downloads.aspx>

Links to manufacturers' SPICE models:

<http://www.intusoft.com/slinks.htm>

Spice Algorithm Overview

<http://www.ecircuitcenter.com/SPICEtopics.htm>

Analogue Simulation Syllabus

This course of lectures deals with simulation of analogue circuits using SPICE and will cover the following topics:

- Text definition of circuits for SPICE, netlists
- DC analysis, transient analysis, frequency response
- Semiconductor device models, sub-circuits
- Schematic entry using Orcad Capture
- Power amplifiers including classes A, B, AB
- Non-linear distortion, power and efficiency
- Buck converters, stabilisation, regulation and efficiency
- Boost and buck/boost converters
- Flyback converters
- Forward converters
- Hysteretic current-control buck converters

Analogue Simulation Prerequisites

You should be familiar with the following topics:

SE1EA5: Electronic Circuits

- Circuit analysis using Kirchhoff's Laws
- Thévenin and Norton's theorems
- The Superposition Theorem
- Semiconductor devices
- Complex impedances
- Frequency response function – gain and phase
- The infinite-gain approximation
- Energy storage and power

SE1EC5: Engineering Mathematics

Analogue Simulation Assessment



EE3H7 Analogue Simulation is assessed by coursework only:

Attendance and laboratory logbook (20% weighting)

A register of attendance will be kept

A 90 minute multi-choice test (40% weighting)

This will be held during the last session of the Autumn Term

A design simulation report (40% weighting)

You will receive details of the design simulation at the end of the Autumn Term and your report must be submitted during the Spring Term

Analogue Simulation Laboratory



You must attend each week - a register will be taken

You will work individually on one of the pcs

You should use the same pc each week

You **must** maintain a logbook containing a record of your work, including:

circuit diagrams

SPICE input files (where appropriate)

simulation outputs and measurements

All work must be dated and be written using a pen (not pencil, except for circuit diagrams)

What is SPICE?

SPICE is an electrical circuit simulation program

It was originally developed at the University of California and was released in 1975 as a Fortran program

Early versions ran in batch mode on main-frame computers with input and output in the form of text files

Recent implementations of SPICE on personal computers tend to be more interactive than the original program

SPICE output is now usually presented in high-resolution graphical form

What is SPICE?

SPICE performs the following types of electrical circuit analysis:

1. DC Analysis - operating point, dc sweep of voltage/current source, dc small-signal sensitivity
2. Transient Analysis - response to time-varying source, Fourier analysis
3. AC Analysis – small-signal frequency response, noise analysis

Session 1

Text definition of circuits for SPICE
SPICE analysis
AC and transient analysis

SPICE Netlist

Circuits are input to the SPICE program as netlists

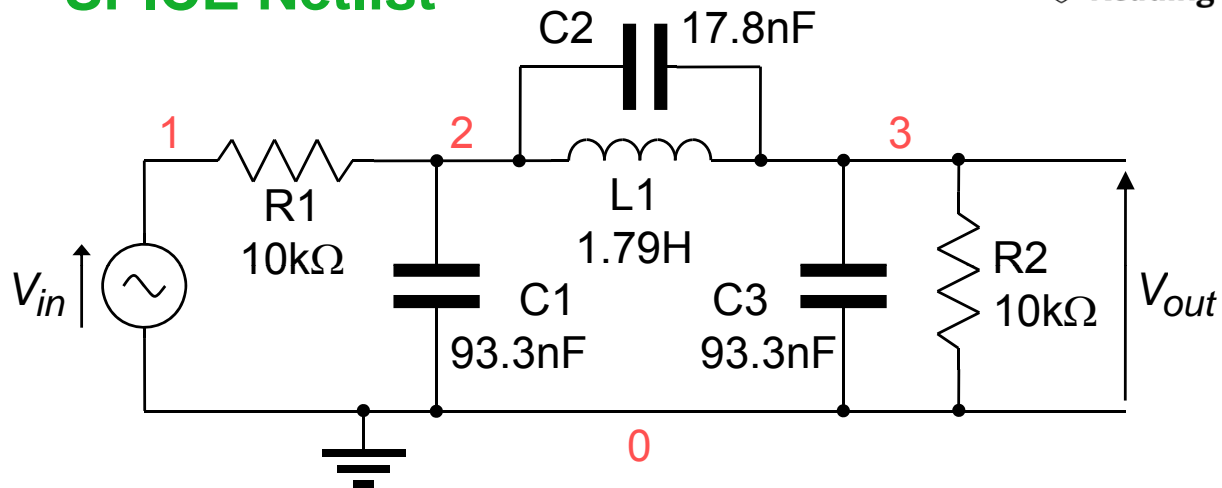
Each point in a circuit where two or more components are connected is a *node*.

Nodes are identified by integer numbers with the ground (or reference) node being assigned 0

A netlist is simply a list of components together with the nodes to which they are connected

In the case of a SPICE netlist the component values are also included in the netlist

SPICE Netlist



Netlist:

R1	1	2	10.0K
R2	3	0	10.0K
C1	2	0	93.3N
C2	2	3	17.8N
C3	3	0	93.3N
L1	2	3	1.79

SPICE Netlist

The first letter of a component symbol determines its type:

Resistors: **R***** n1 n2 value**

Capacitors: **C***** n1 n2 value**

Inductors: **L***** n1 n2 value**

Connection nodes are specified by integers *n1* and *n2*.

Component values are given as floating-point numbers

Units are Ohm for Resistors, Farad for capacitors and Henry for Inductors

SPICE Netlist

Scaling factors for component values:

T	10^{12}
G	10^9
Meg	10^6
K	10^3
m	10^{-3}
u	10^{-6}
n	10^{-9}
p	10^{-12}
f	10^{-15}

Thus the following all represent the same component value:

```
Cp 1 2 0.00000047
Cp 1 2 4.7E-7
Cp 1 2 0.47u
Cp 1 2 470.0n
Cp 1 2 470.0nF
```

SPICE Netlist

Independent voltage or current sources are used for signal inputs (and power supplies):

```
V***** n1 n2 <<DC>dcval > <<AC<acmag<acphase>>>
I***** n1 n2 <<DC>dcval > <<AC<acmag<acphase>>>
```

An ac analysis is specified by:

```
.AC type np fstart fstop
```

type should be either *OCT* (octave) or *DEC* (decade)

np is the number of points per octave or decade

fstart and *fstop* are the initial and final frequencies

SPICE Netlist

To obtain a printout of the response a print statement is used:

```
.PRINT prtype ov1, <ov2 .. <ov8>>
```

prtype specifies the type of analysis and can be either *AC*, *DC* or *TRAN* (transient)

ov1 .. *ov8* are the output voltages and have the general form:

```
V(n1<, n2>)
```

The voltage between *n1* and *n2* is printed; if the second node *n2* is omitted then the reference node 0 is used by default

It is also possible to specify current outputs

SPICE Netlist

In ac analysis the magnitude of the ac voltage is printed by default

Other properties of the outputs, such as real part or phase can also be specified:

Real part	VR(n1<, n2>)
Imaginary part	VI (n1<, n2>)
Magnitude	VM(n1<, n2>)
Phase	VP(n1<, n2>)
Decibels	VDB(n1<, n2>)

To print the voltage in decibels and the phase at the output node 3 of the filter:

```
.PRINT AC VDB(3) VP(3)
```

Complete SPICE Input File

A SPICE input file must start with a title line, finish with a .END line and may contain comment lines (starting with *):

```
Elliptic Filter
* 3rd-order low-pass filter
* Implemented as equally-terminated ladder
R1  1  2  10.0K
R2  3  0  10.0K
C1  2  0  93.3N
C2  2  3  17.8N
C3  3  0  93.3N
L1  2  3  1.79
VIN 1  0  AC  1V
.AC  DEC  8  30HZ  3KHZ
.PRINT AC VDB(3)  VP(3)
.END
```


AC Analysis Using PSpice

From the Windows desktop select: Start\Run, type: notepad and select: OK

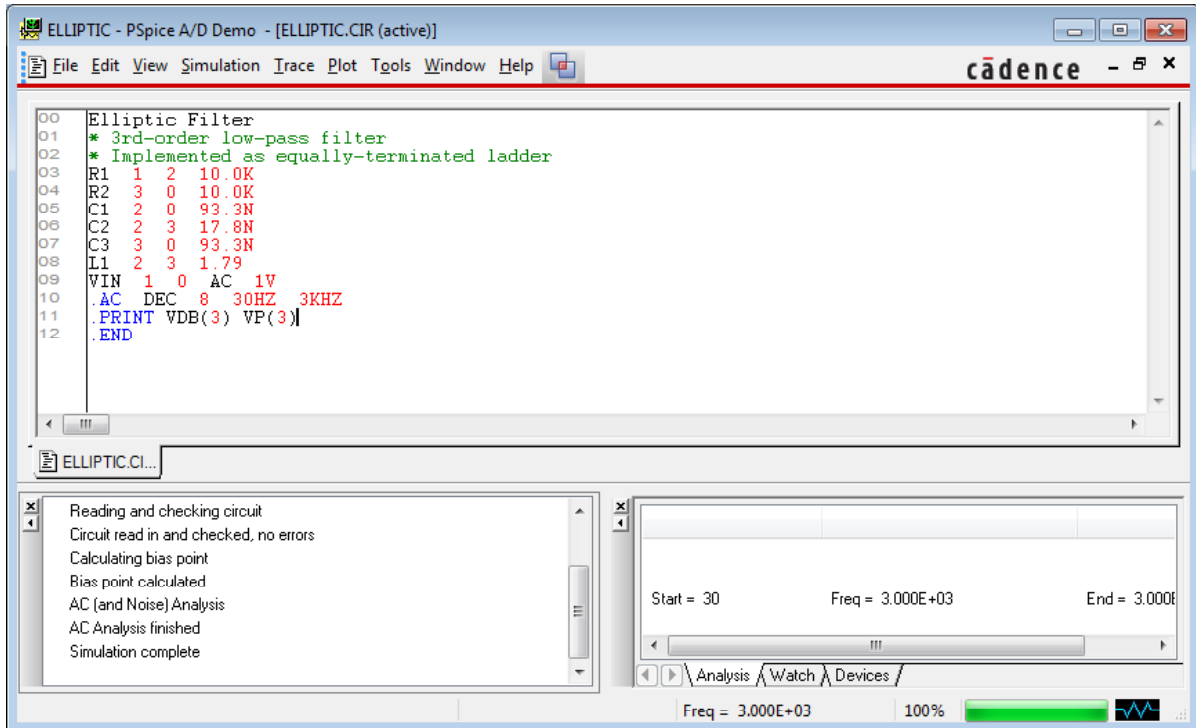
Now enter the SPICE netlist of the filter and save it to a file in your directory: elliptic.cir

Click on: Start / All Programs / Cadence / Release 16.3 / PSpice AD; this will bring up the PSpice GUI

Select: File\Open, then: Files of type: Circuit Files, finally locate your SPICE netlist file

Now select: Simulation/Run elliptic or click the icon:  - the simulation should run without error

AC Analysis Using PSpice



AC Analysis Using PSpice

The PSpice simulator will have generated an output file: elliptic.out

Select: View/Output file:

FREQ	VDB(3)	VP(3)
3.000E+01	-6.128E+00	-1.094E+01
4.001E+01	-6.209E+00	-1.450E+01
5.335E+01	-6.348E+00	-1.911E+01
7.114E+01	-6.577E+00	-2.498E+01
9.487E+01	-6.940E+00	-3.226E+01
1.265E+02	-7.469E+00	-4.095E+01
1.687E+02	-8.145E+00	-5.090E+01
2.250E+02	-8.808E+00	-6.216E+01
.
2.250E+03	-4.074E+01	-8.238E+01
3.000E+03	-4.271E+01	-8.433E+01

AC Analysis Using PSpice

To obtain a graphical display of the response it is necessary to include a .PROBE line in the netlist:

```
. . . . .  
L1 2 3 1.79  
VIN 1 0 AC 1V  
.AC DEC 100 30HZ 3KHZ  
.PROBE  
.END
```

Note that the number of points per decade has been increased to 100 to give a smooth curve

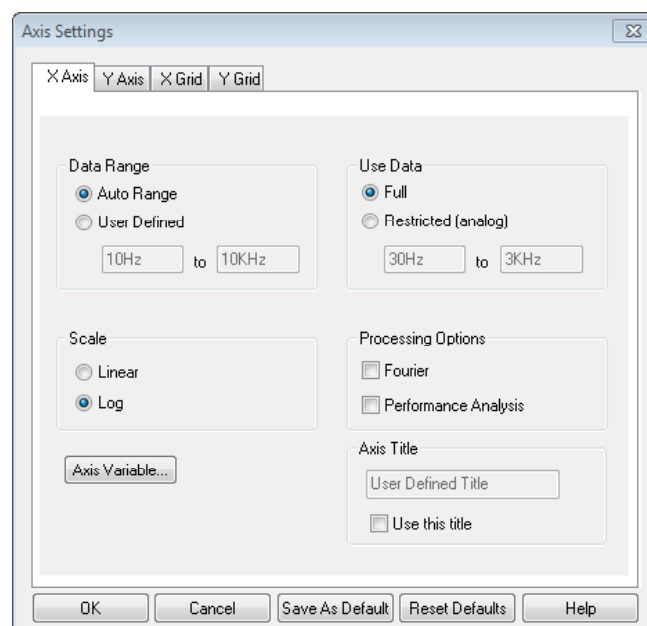
Now run the simulation again - a plot window should appear within the PSpice GUI

Before the response can be viewed it is necessary to configure the plot

AC Analysis Using PSpice

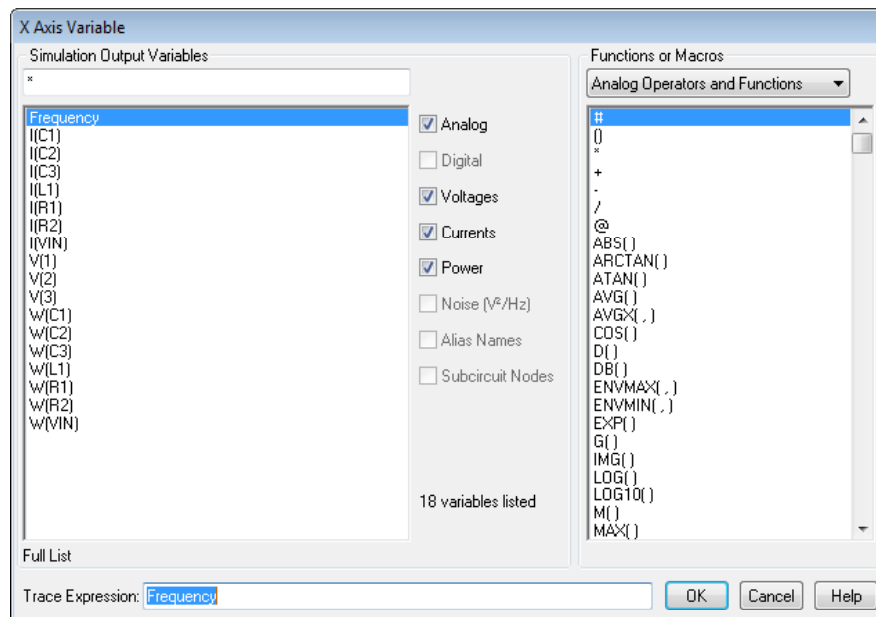
Click on: Plot/Axis Settings and select X Axis tab

Under Data Range select: Auto Range and under Use Data select: Full



AC Analysis Using PSpice

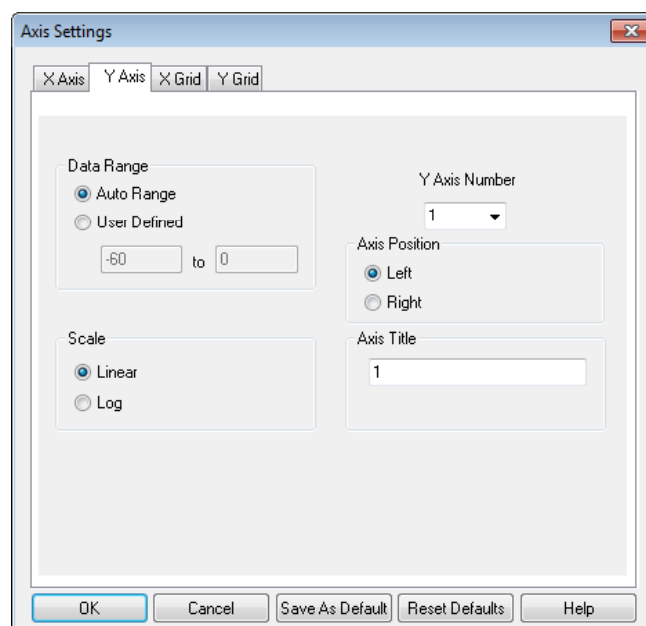
Click: Axis Variable and select: Frequency from the table and click OK



AC Analysis Using PSpice

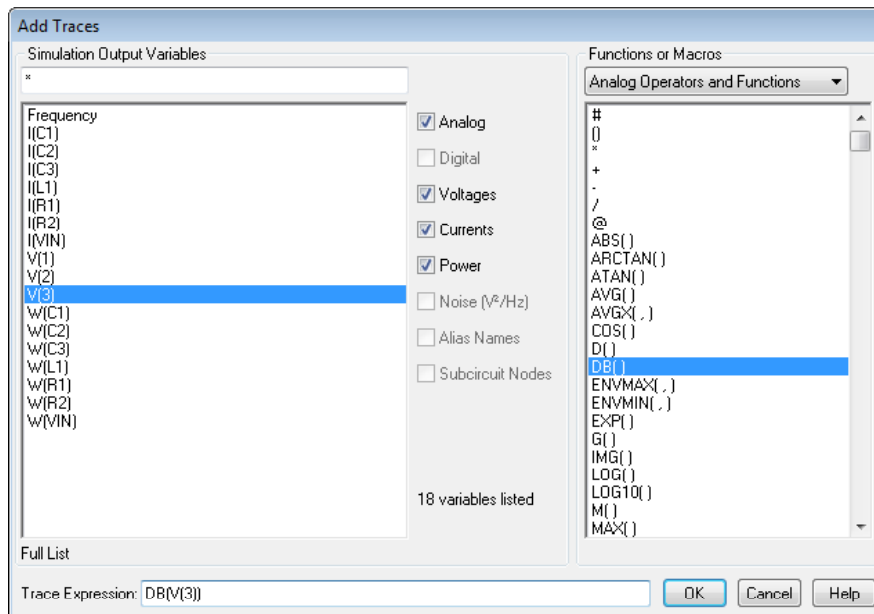
Select Y Axis tab

Under Data Range select: Auto Range and click: OK

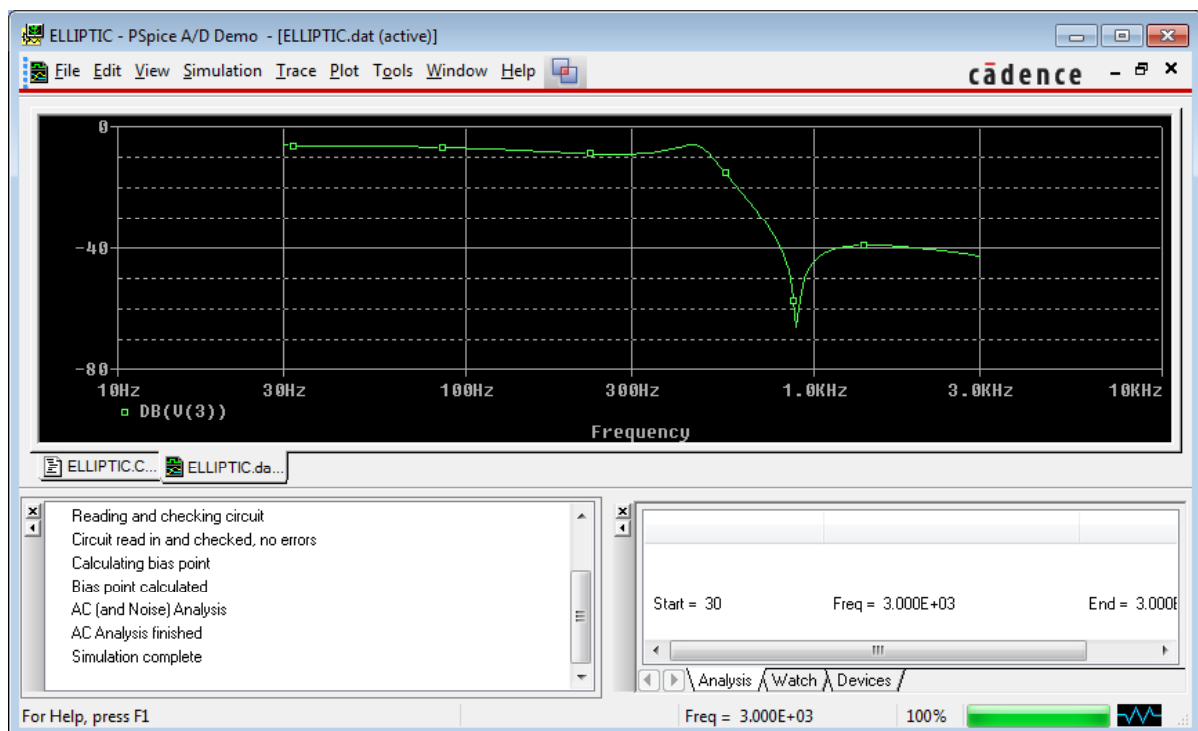


AC Analysis Using PSpice

Click on Trace/Add Trace, in right-hand pane select: DB(), in left-hand pane select: V(3) and click: OK

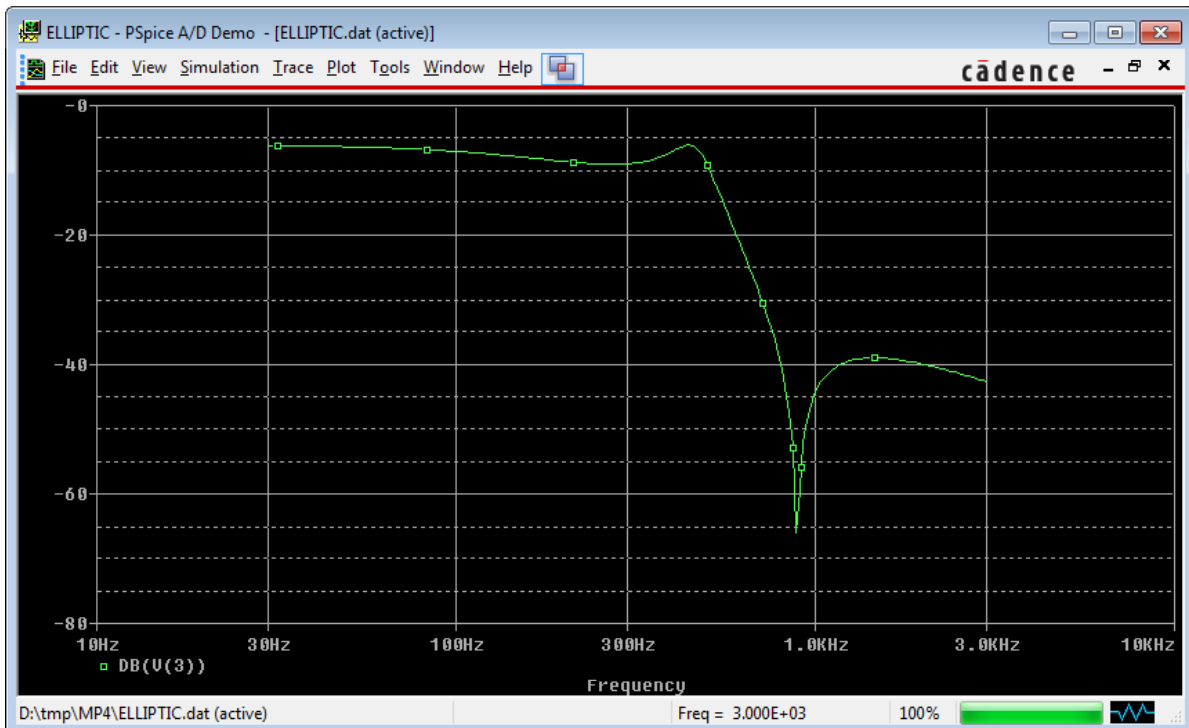


AC Analysis Using PSpice



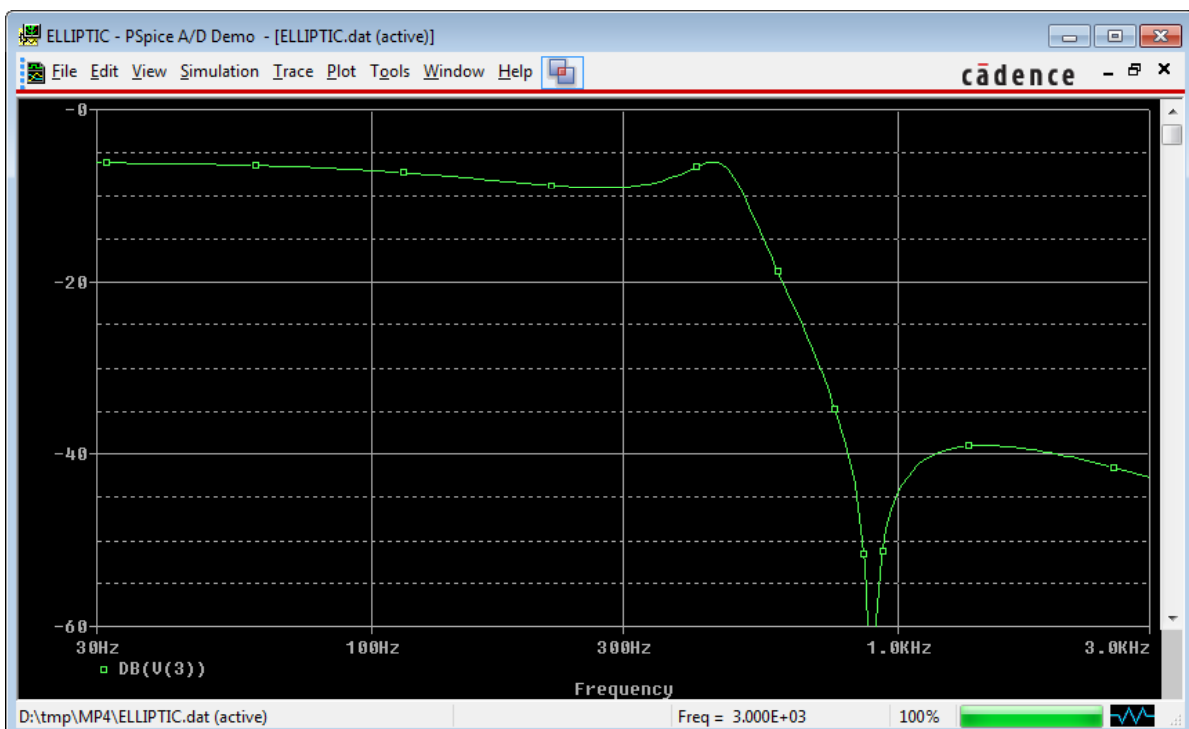
AC Analysis Using PSpice

To obtain a clearer plot click the Alternate Display icon:



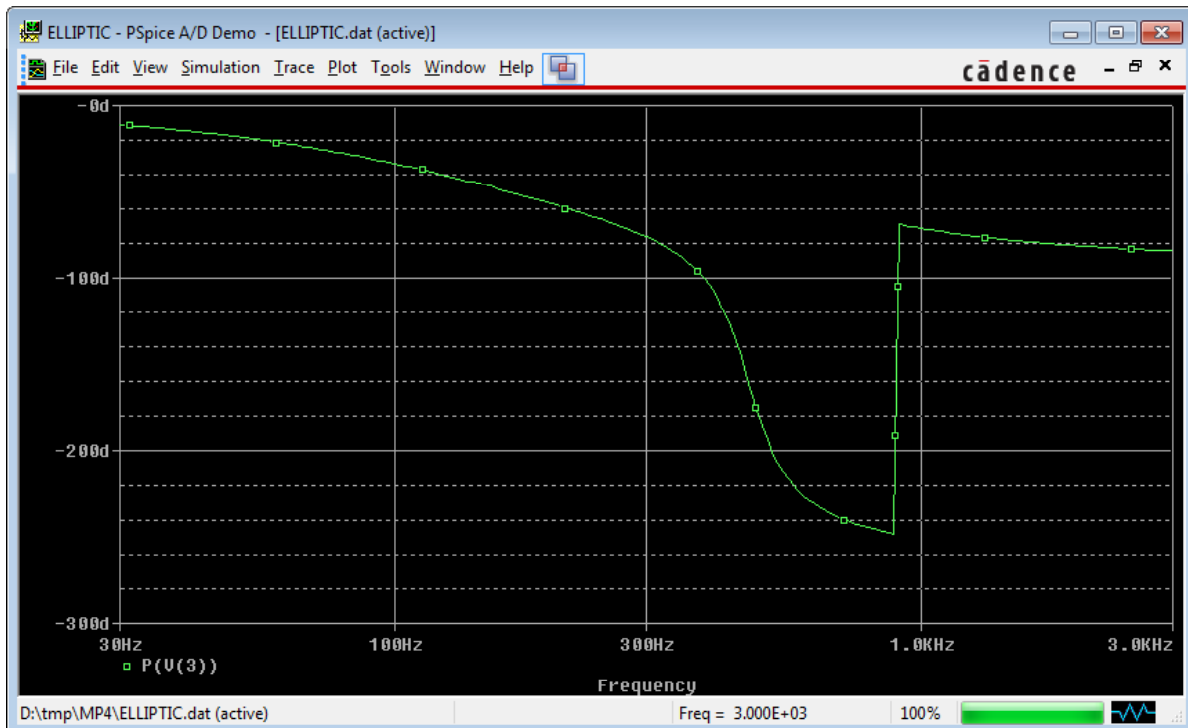
AC Analysis Using PSpice

Change X range to 30 Hz → 3 kHz and Y range to -60 → 0



AC Analysis Using PSpice

Change the trace to the output phase: $P(V(3))$



Measurements

Return to the plot of gain in dB against frequency

Click on: Trace / Cursor / Display

Now click the mouse close to the trace

The cross-hairs will centre on the trace and a Probe Cursor window will give the X and Y values

Measure the maximum and minimum gain in the pass-band; hence determine the pass-band ripple

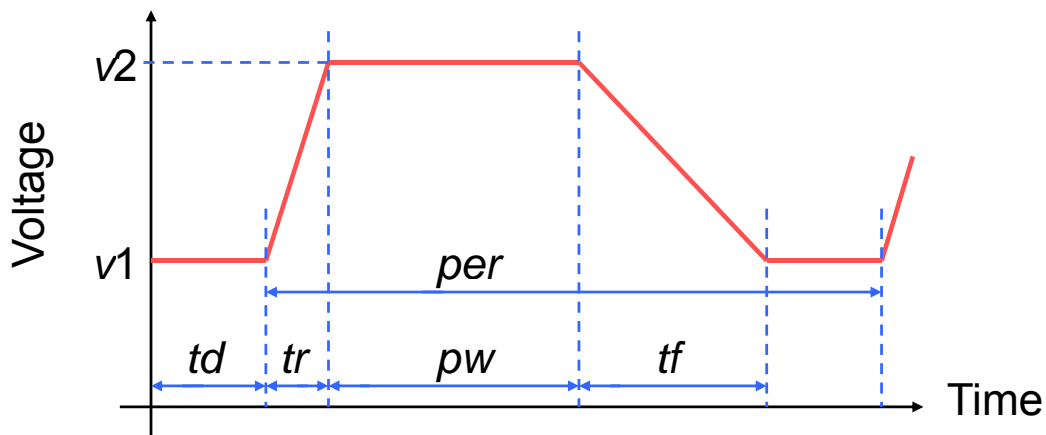
Measure the maximum gain in the stop-band

Transient Analysis Using PSpice

Input waveforms can be dc, ac, pulse, sinusoidal, exponential or piece-wise linear

A pulse waveform is specified:

```
PULSE ( v1 v2 <td <tr <tf <pw <per>>>> )
```



Transient Analysis Using PSpice

A unit-step input voltage on node 1 can be specified by:

```
VIN 1 0 PULSE ( 0.0 1.0 0.0 )
```

A transient analysis is specified by:

```
.TRAN tstep tstop <tstart <tmax>> <UIC>
```

tstart, *tstop* and *tstep* are the are the start, stop and time step of the display

The parameter *tmax* is only used if the calculation time is required to be less than the display time step

UIC (use initial conditions) causes the analysis to start from specified values, rather than from an initial dc analysis

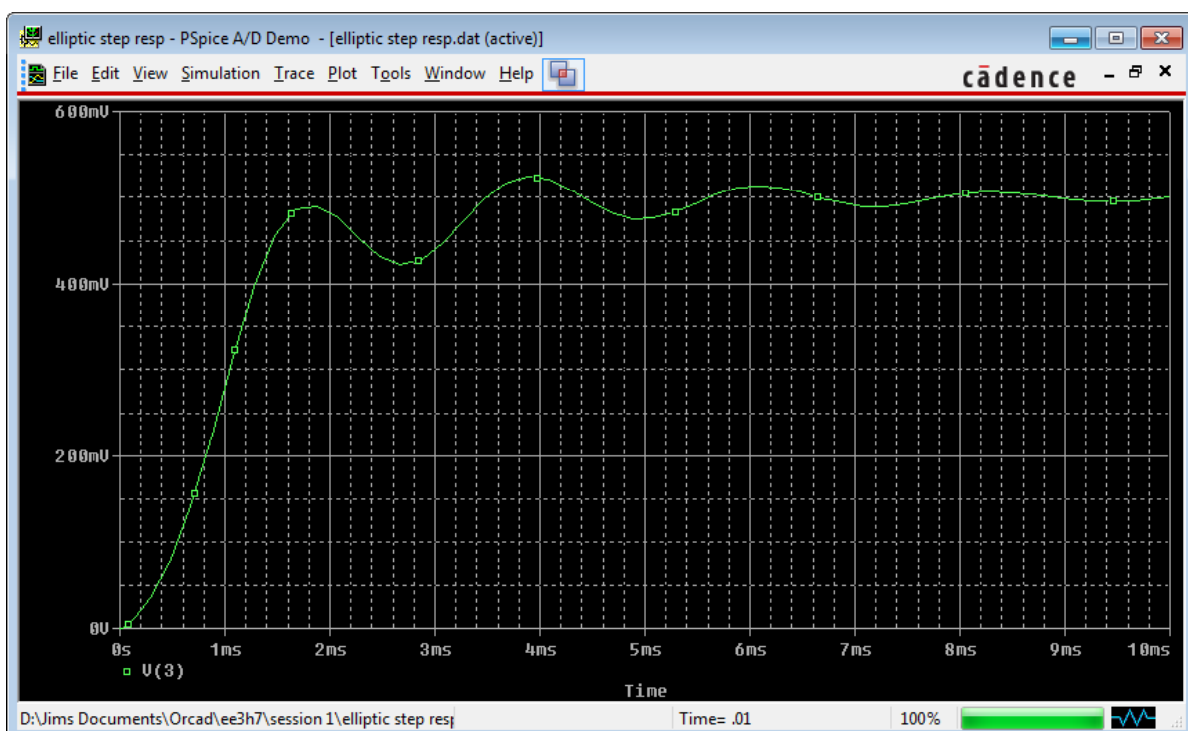
Transient Analysis Using PSpice

The complete PSpice input file for generating a unit-step response is therefore:

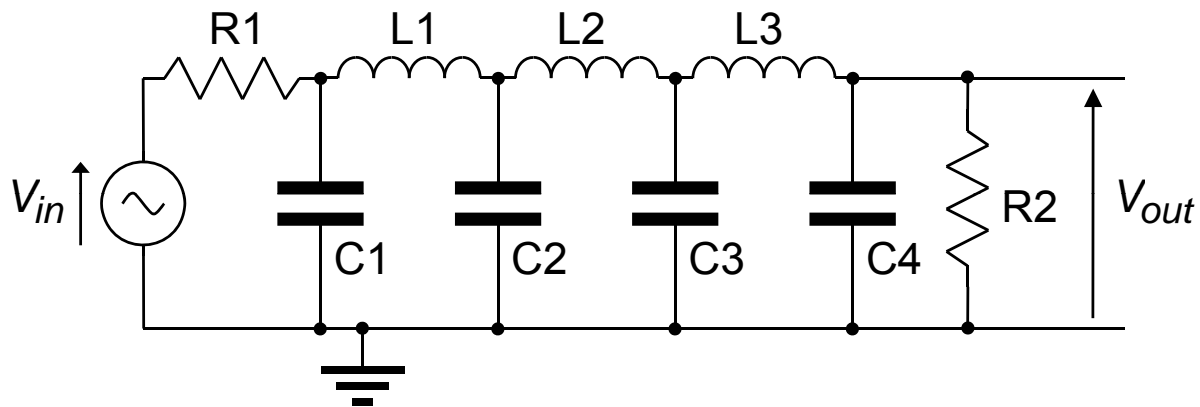
```
Elliptic Filter
* 3rd-order low-pass filter
* Implemented as equally-terminated ladder
R1 1 2 10.0K
R2 3 0 10.0K
C1 2 0 93.3N
C2 2 3 17.8N
C3 3 0 93.3N
L1 2 3 1.79
VIN 1 0 PULSE ( 0.0 1.0 0.0 )
.TRAN 0.0001 0.01
.PROBE
.END
```

Transient Analysis Using PSpice

Run simulation and add trace: V(3)



Bessel Low-Pass Filter



$$R1 = 1 \text{ k}\Omega$$

$$R2 = 1 \text{ k}\Omega$$

$$L1 = 5.1869 \text{ mH}$$

$$L2 = 11.173 \text{ mH}$$

$$L3 = 17.590 \text{ mH}$$

$$C1 = 1.7603 \text{ nF}$$

$$C2 = 8.3540 \text{ nF}$$

$$C3 = 13.831 \text{ nF}$$

$$C4 = 36.063 \text{ nF}$$

Bessel Low-Pass Filter

Write a SPICE input file for the Bessel Filter and perform an ac analysis over the frequency range 1 kHz to 100 kHz

Plot the gain against frequency and determine the frequency at which the gain is -80 dB

Plot the phase against frequency and determine the phase shift at 10 kHz

Perform a transient analysis with a unit step input over the time range 0 to 200 μ s in steps of 1 μ s

Plot the unit step response and determine the time for the output to reach 90% of its final value

Session 2

Device models
Linear dependent sources
Sub-circuits

Semiconductor devices

SPICE recognises four different types of semiconductor device:

- junction diode
- bipolar transistor (BJT)
- junction field-effect transistor (JFET)
- insulated-gate field-effect transistor (MOSFET)

Semiconductor devices have complex characteristics and are characterised by a number of parameters

A device model is required for each semiconductor device in the circuit

The device models are usually imported from a model library

Junction Diode

The element description for a junction diode is of the form:

```
DXXXXXX na nc mname <AREA=aval > <OFF> <I C=vd>
```

Nodes *na* and *nc* are the anode and cathode nodes

mname is the device model, which must be included in the SPICE input file

AREA is an optional parameter (default value 1.0) which determines the effective number of parallel devices

OFF sets the starting conditions to off for this device during dc analysis

Junction Diode

Junction diode example:

```
DBRIDGE 2 4 D1N4002
```

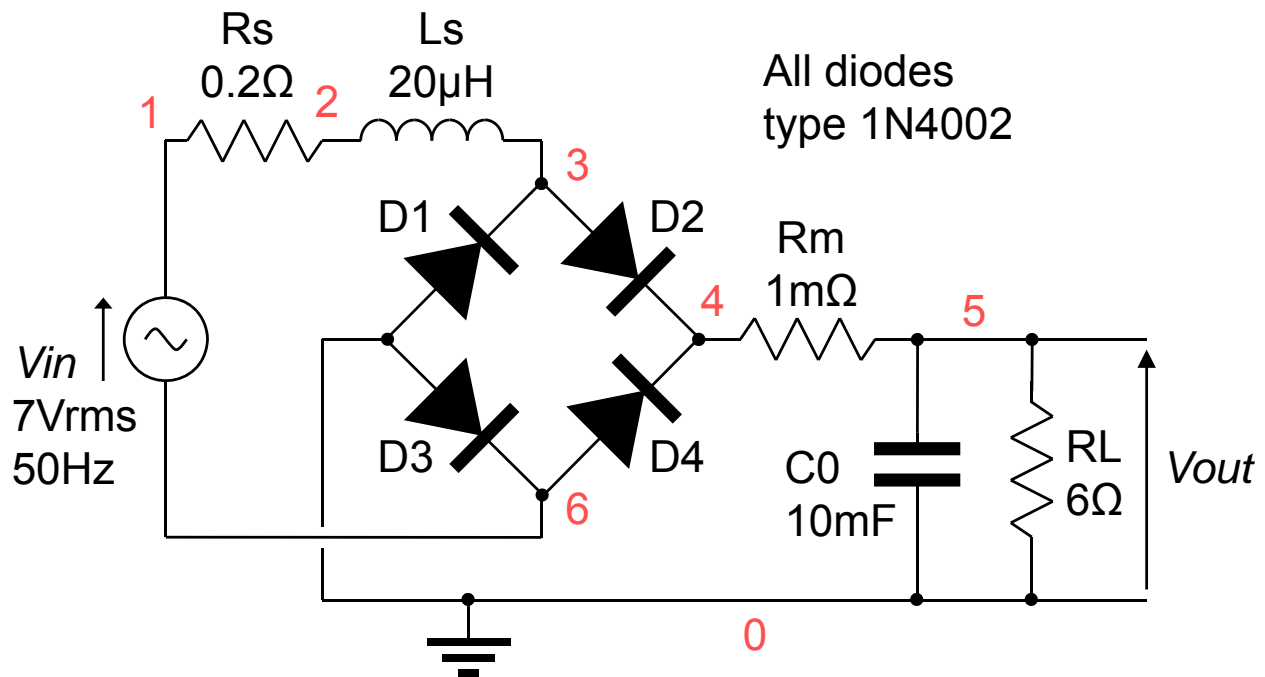
An appropriate device model would be:

```
** 1N4002 General purpose rectifier diode  
.MODEL D1N4002 D (IS=7.0E-10 RS=0.1 N=1.6  
                TT=1.0E-7 CJO=4.0E-11  
                BV=100.0)
```

This model might form part of the SPICE input file, or if it is in a library file *Lib.cir* it can be imported:

```
.LIB Lib.cir
```

Full-Wave Rectifier



Full-Wave Rectifier

SPICE circuit description of full-wave rectifier

Diode model D1N4002 is imported from Lib.cir

Note that the input is a dc voltage of 5V

Bridge Rectifier

```
.LIB Lib.cir
RS 1 2 0.2
LS 2 3 20.0U
D1 0 3 D1N4002
D2 3 4 D1N4002
D3 0 6 D1N4002
D4 6 4 D1N4002
RM 4 5 1.0M
C0 5 0 10000UF
RL 5 0 6.00HM
VIN 1 6 DC 5.0
.END
```

Full-Wave Rectifier

Use Notepad to create Rectifier.cir, extract Lib.cir from spice_library.zip and run the simulation

Because no analysis type is specified PSpice will perform a dc analysis only

Inspect the output file and determine the voltage across the load resistor R_L (node 5)

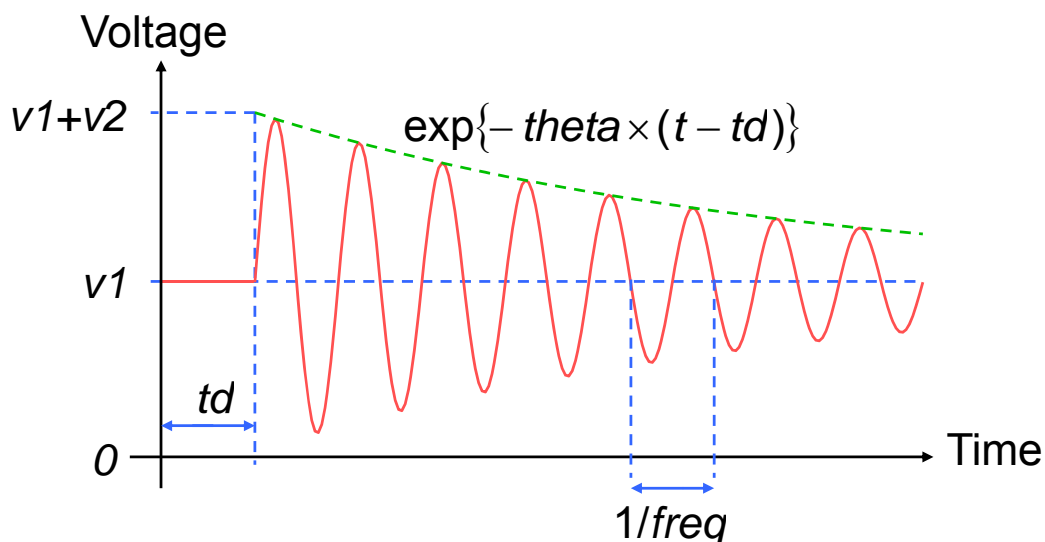
Change the input voltage V_{in} to -5V and again determine the voltage across resistor R_L

Calculate the voltage drop across the conducting diodes

Transient Analysis Using PSpice

A sinusoidal input waveform is specified by:

`SIN (v1 v2 freq <td <theta>>)`



Full-Wave Rectifier

Assume that the input to the rectifier is a 7V rms mains-frequency sinusoid

Use the following lines to specify a transient analysis with a sinusoidal input:

```
VIN 1 6 SIN ( 0.0V 9.8V 50HZ )  
.TRAN 0.5MS 80.0MS 0.0MS 0.1MS
```

Include a .PROBE line to obtain graphical output

Run the simulation and display the output voltage $V(5)$ and the current through the diode bridge: $I(Rm)$

Full-Wave Rectifier

Measure the dc output voltage after the circuit has settled down and the repetitive peak current in the diode bridge

The specification for this rectifier circuit requires that the output ripple be less than 0.2V

Change the component values to meet this specification

Remember that capacitor values are normally in the E3 series: 1, 2.2, 4.7, 10 etc

What effect does this have on the repetitive peak current?

Linear Dependent Sources

SPICE recognises four different types of linear dependent source:

- voltage-controlled voltage source (VCVS)
- voltage-controlled current source (VCCS)
- current-controlled voltage source (CCVS)
- current-controlled current source (CCCS)

Element descriptions are, respectively:

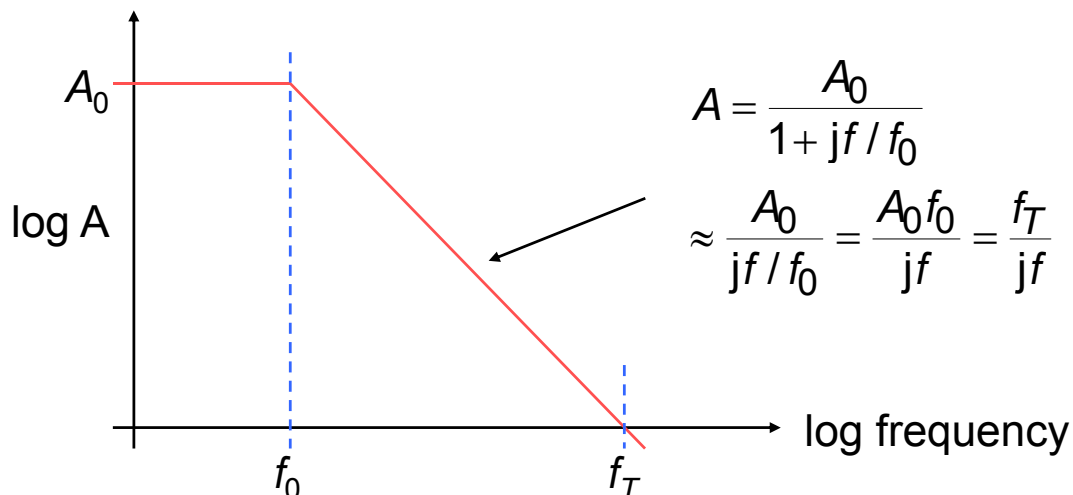
```
E***** n1 n2 n3 n4 value
G***** n1 n2 n3 n4 value
H***** n1 n2 vnam value
F***** n1 n2 vnam value
```

Output nodes are $n1, n2$

Input is nodes $n3, n4$ or the current through a voltage source

Operational Amplifiers

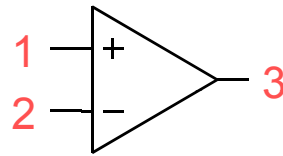
Open loop frequency response of an operational amplifier:



A typical operational amplifier (TL071) has a gain-bandwidth product $fT = 3\text{MHz}$

Operational Amplifiers

A high-gain voltage-controlled voltage source (VCVS) can be used to represent an operational amplifier



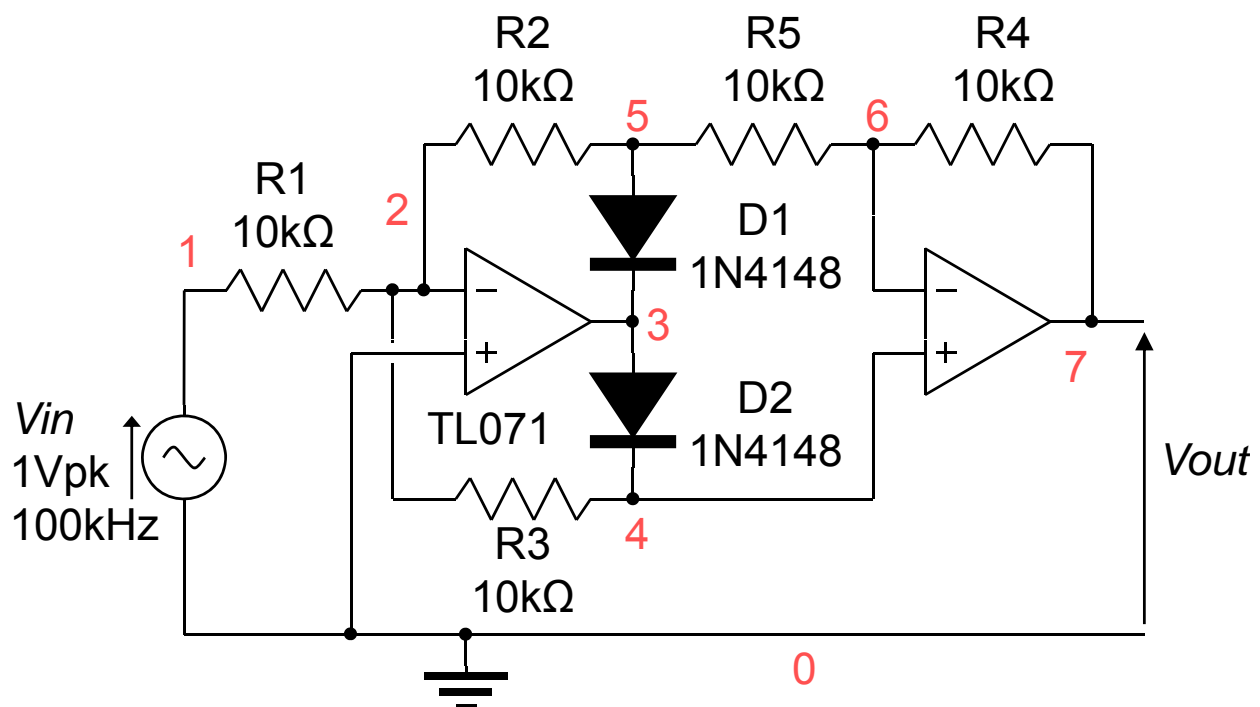
For example at 100kHz a TL071 operational amplifier (gain-bandwidth product $f_T = 3\text{MHz}$) has a voltage gain of 30:

E1 3 0 1 2 30

Operational amplifiers have a single output node

(The other output of the dependent source is implicitly connected to ground)

Precision Rectifier



Precision Rectifier

Write a SPICE circuit description file for the precision rectifier, importing Lib.cir for the 1N4148 diode model (D1N4148)

Use the following dependent source lines for the operational amplifiers:

```
E1 3 0 0 2 30
E2 7 0 4 6 30
```

Perform a transient analysis over the time range 0 to 20 μ s:

```
.TRAN 0.1US 20.0US 0.0US 0.01US
```

Plot the voltages on nodes 1 and 7 and measure the peak output voltage for both positive and negative input half cycles

Sub-Circuits

Circuits often contain a number of identical elements such as operational amplifiers or logic gates

A sub-circuit can be defined which is then incorporated in the circuit description by a single line sub-circuit call

A sub-circuit definition has the following form:

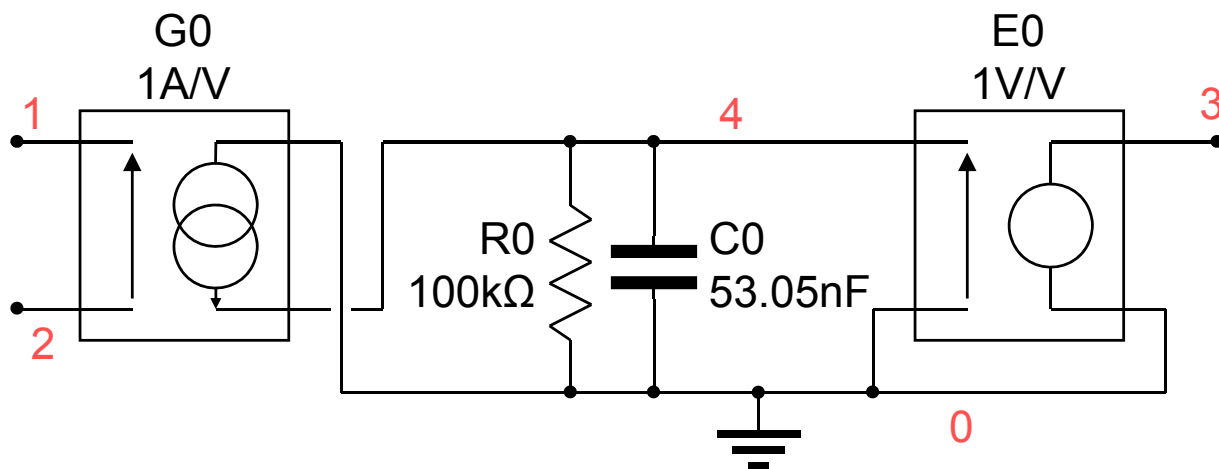
```
.SUBCKT subname n1 <n2 <n3 .. >>
[defi ni ti on]
.ENDS
```

A sub-circuit call has the following form:

```
X***** n1 <n2 <n3 .. >> subname
```

Operational Amplifiers

Linear operational amplifier model (TL071):



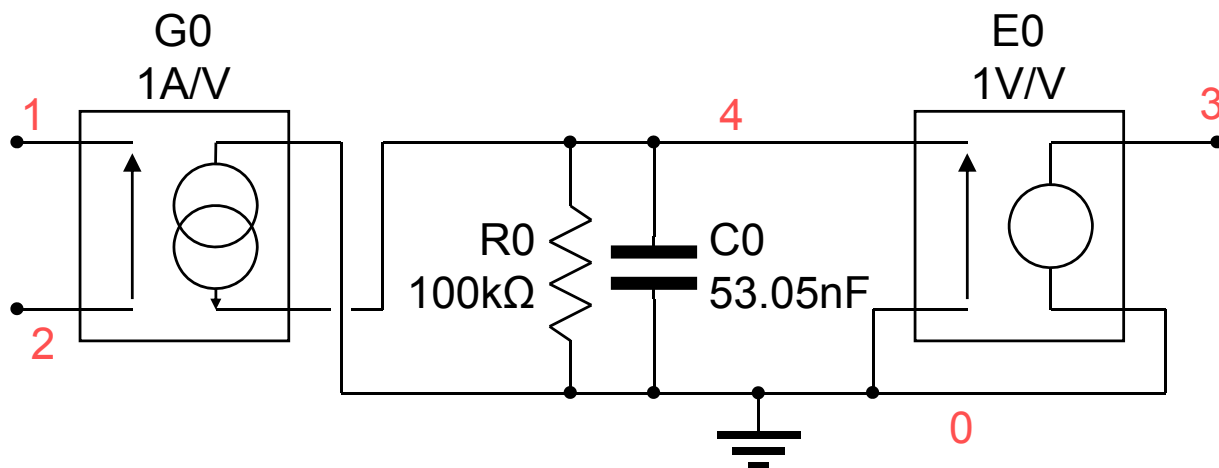
$$f_T = \frac{E0 \times G0}{2\pi \times C0}$$

$$= 3 \text{ MHz}$$

$$A_0 = E0 \times R0 \times G0$$

$$= 100000$$

Operational Amplifiers



```
.SUBCKT MYTL071 1 2 3
GO 0 4 1 2 1.0
R0 4 0 100K
C0 4 0 53.05n
E0 3 0 4 0 1.0
.ENDS MYTL071
```

Operational Amplifiers

Edit your circuit description file to include the TL071 sub-circuit

Use the following sub-circuit lines for the operational amplifiers:

```
X1 0 2 3 MYTL071
X2 4 6 7 MYTL071
```

Perform a transient analysis over the time range 0 to 20 μ s

Plot the voltages on nodes 1 and 7 and measure the peak output voltage for both positive and negative input half cycles

Operational Amplifiers

More complete operational amplifier models can be obtained from the manufacturers

In the library file Lib.cir there is a model of the TL071:

```
* TL071 OPERATIONAL AMPLIFIER * CREATED USING
* CONNECTIONS:      NON-INVERTING INPUT
*                   | INVERTING INPUT
*                   | | POSITIVE POWER SUPPLY
*                   | | | NEGATIVE POWER SUPPLY
*                   | | | | OUTPUT
*                   | | | | |
.SUBCKT TL071      1 2 3 4 5
C1      11 12 3.498E-12
. . . . .
.ENDS
```

Operational Amplifiers

Edit your circuit description file to include positive and negative dc power supplies:

```
VS1 8 0 DC 12V
VS2 9 0 DC -12V
```

Use the following sub-circuit lines for the operational amplifiers:

```
X1 0 2 8 9 3 TL071
X2 4 6 8 9 7 TL071
```

Perform a transient analysis over the time range 0 to 20 μ s

Plot the voltages on nodes 1 and 7 and measure the peak output voltage for both positive and negative input half cycles

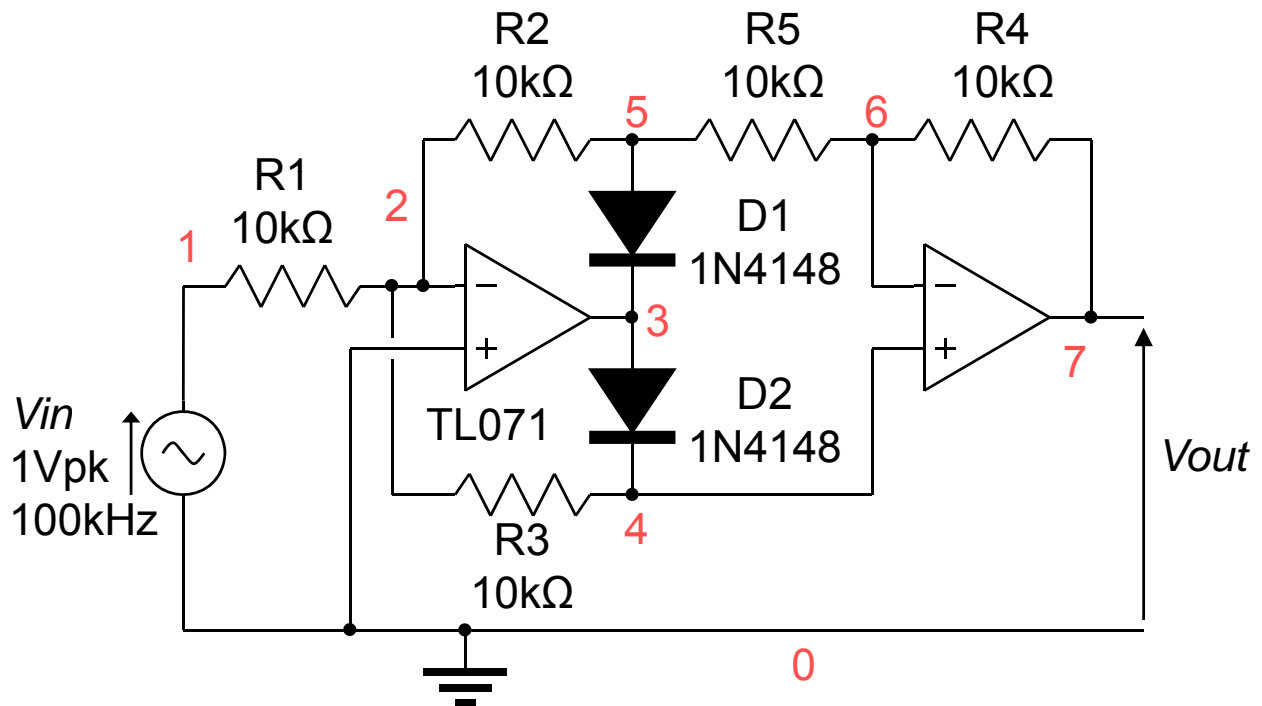
Analogue Simulation

Session 3

Schematic Entry
Precision Rectifier
Elliptic Filter

Schematic Entry

Schematic of precision rectifier:



Schematic Entry

Click on: Start / All Programs / Cadence / Release 16.3 / ORCAD Capture CIS

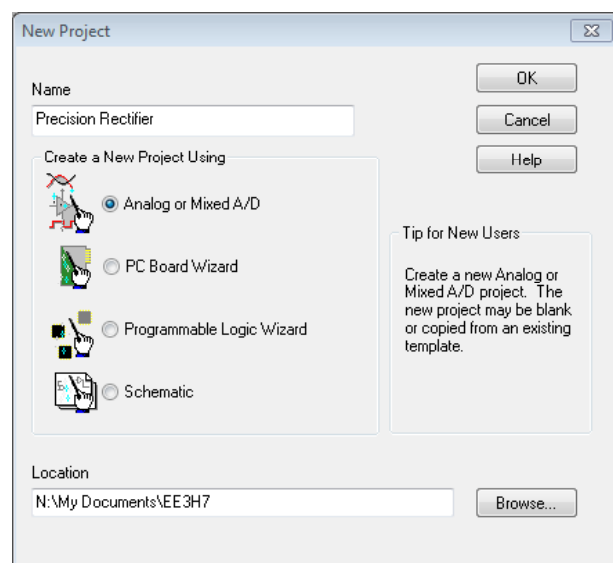
Select: File / New / Project

Enter name: Precision Rectifier

Select: Analog or Mixed A/D

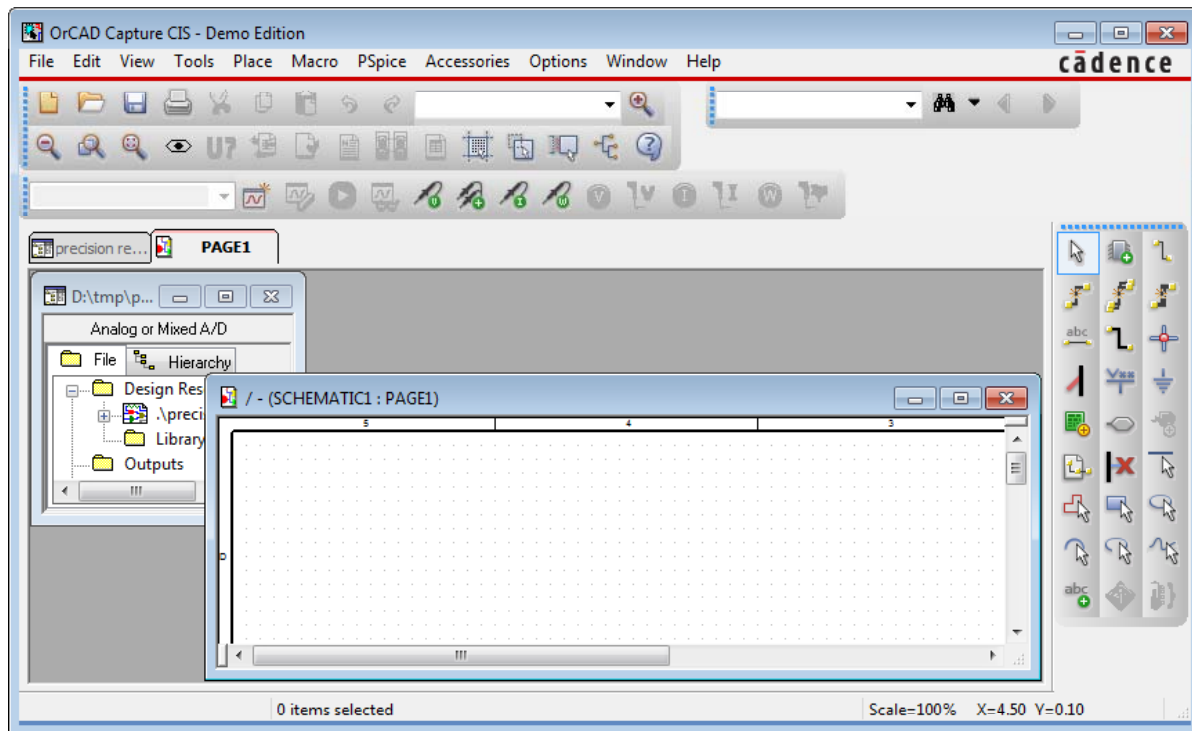
Browse to your directory on the N: drive

Click: OK



A Create PSpice Project window will appear; select: create a blank project and click: OK

Schematic Entry



Schematic Entry

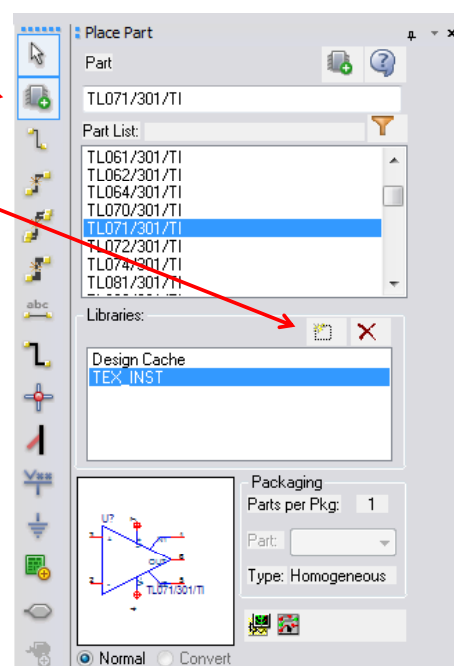
Click on: Place Part icon of the right-hand tool palette:

Click on: Add Library icon:

Add the following PSpice library files:

analog.olb
diode.olb
source.olb
tex_inst.olb

From TEX_INST select:
TL071/301/TI and click: OK



Schematic Entry

Move cursor over schematic window - use H, V or R to flip horizontally, vertically or rotate - finally click to place

Similarly select and place:

From DIODE select: D1N4148/27C (place 2 items)

From ANALOG select: R (place 5 items)

From SOURCE select: VSIN (input voltage)

From SOURCE select: VDC (power supplies for op-amps)

Click on the resistor values and set to 10k

Click on the input voltage VOFF and set to 0

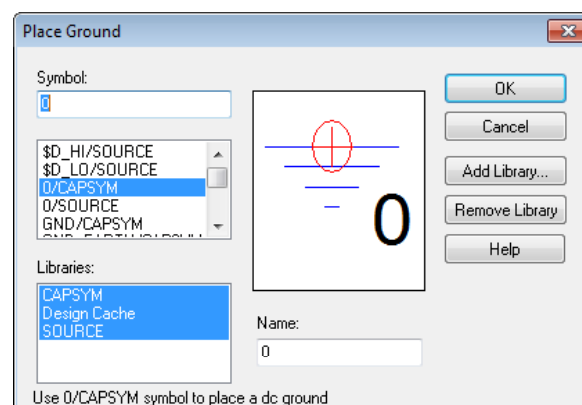
Click on the input voltage VAMPL and set to 1.0

Click on the input voltage FREQ and set to 100kHz

Click on the power supply voltages and set to 12V

Schematic Entry

Click on: Place Ground icon of the tool palette:



In the Place Ground window select: 0/CAPSYM and click: OK

Move cursor over schematic and place ground

Schematic Entry

Click on: Place Wire icon of the tool palette: 

Connect parts to create circuit diagram

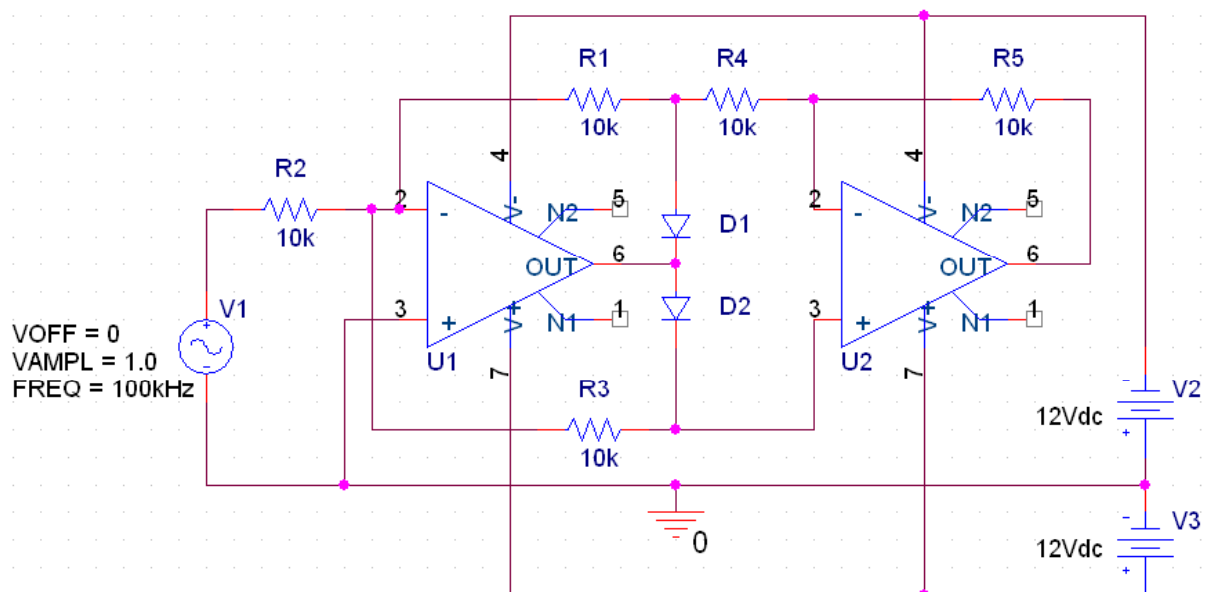
Leave null pins (*N1* and *N2*) on op-amps disconnected

Connect power pins (*V+* and *V-*) on op-amps to $\pm 12\text{V}$ power supplies

Carefully check schematic against correct circuit diagram

Schematic Entry

Complete schematic:



Simulation

Click on: PSpice / New Simulation Profile and enter name: time-domain response, then click: Create

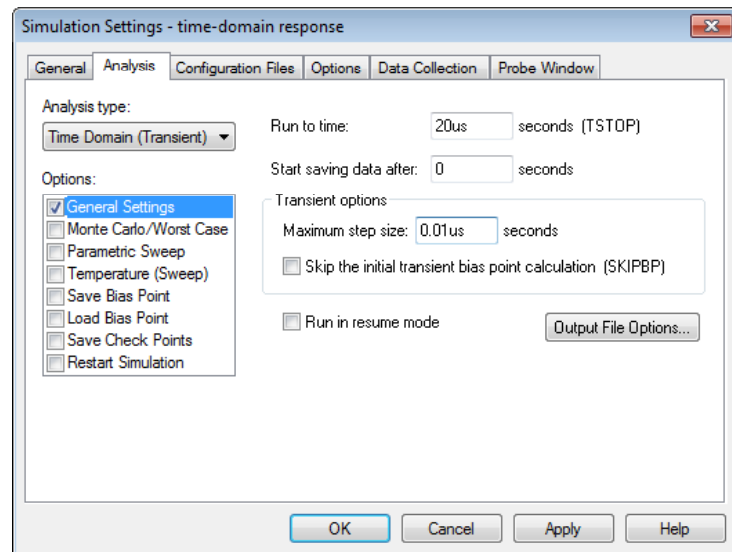
Under analysis tab set:

Analysis type =
Time Domain

Run to Time =
20us

Maximum step
size = 0.01us

Click: OK



Simulation

Click: Run Simulation icon: 

This should complete without errors and a PSpice graphical output window should appear

Traces can be added in the usual way but it is more convenient to use markers

Click on: Voltage Marker icon: 

Place voltage markers on the input and the output

Run the simulation - the output window should now display the input and output voltages of the precision rectifier

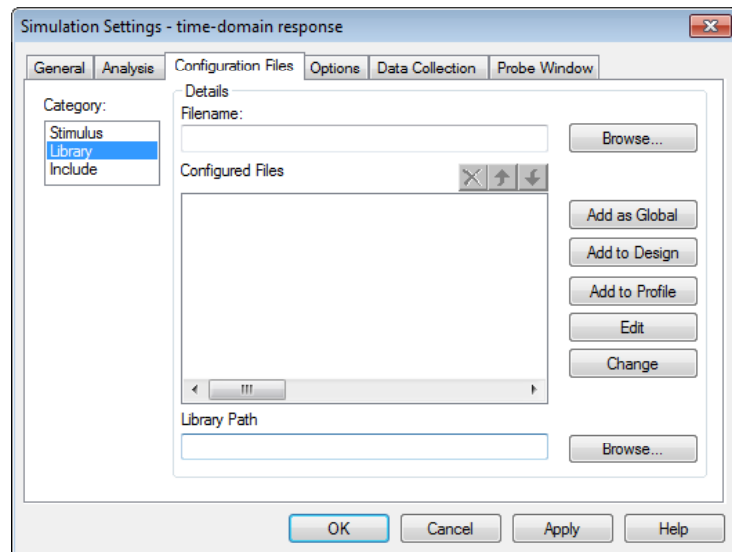
Schematic Entry

If on running the simulation an error message concerning missing device libraries is obtained:

Click on: PSpice / Edit Simulation Profile

Select the tab: Configuration Files

In Category select: Library



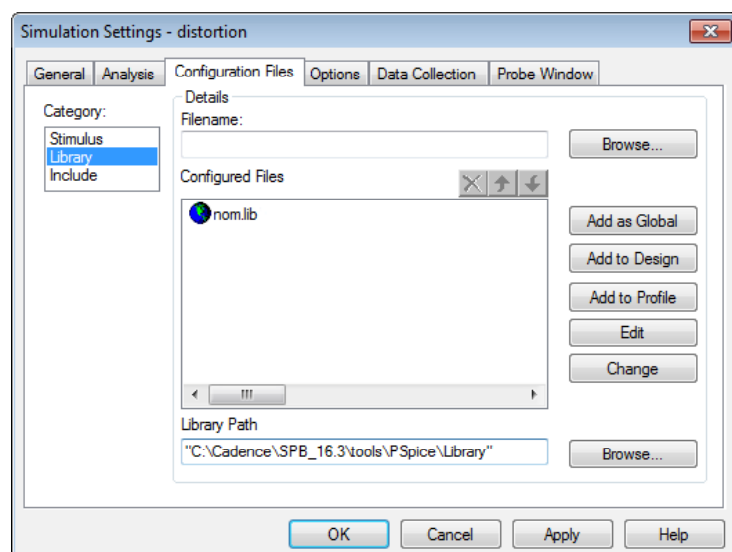
Schematic Entry

Browse to C:\CADENCE \ SPB_16.3 \ tools \ PSpice \ library and select file: nom.lib

Now click on: Add as Global

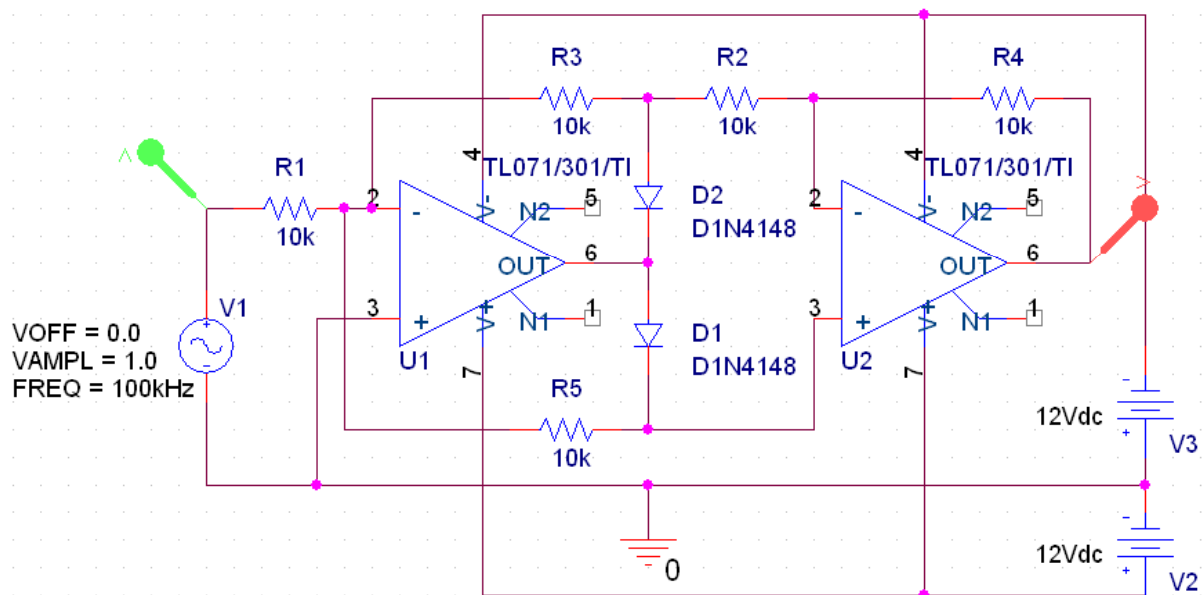
Click on: Apply, then: OK

Run simulation again



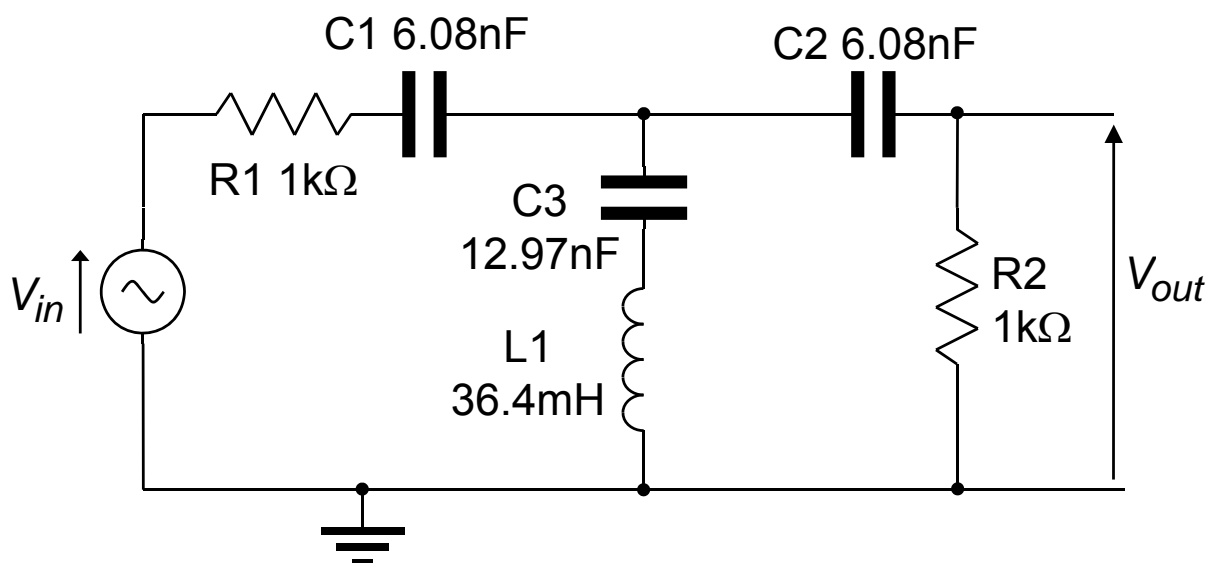
Simulation

Complete schematic with voltage markers:



Elliptic Filter

Passive high-pass elliptic filter:



Elliptic Filter

Input the schematic of the elliptic filter amplifier

From SOURCE select: VAC for the input voltage, and set ac to 1.0V and dc to 0.0v

Simulate the ac response between 100Hz and 1MHz

Although this is a perfectly correct and usable circuit PSpice will generate an error message in the PSpice output file:

```
ERROR -- Node Nxxxxx is floating
```

This is because there is no dc path from the node connecting C1, C2 and C3

PSpice cannot therefore establish the dc bias at this node

Elliptic Filter

PSpice
output file:

```
*Analysis directives:
.AC DEC 200 10 10k
.PROBE V(alias(*)) I(alias(*))
.INC "..\SCHEMATIC1.net"
**** INCLUDING SCHEMATIC1.net ****
* source PASSIVE
V_V1          N00269 0 DC 0Vdc AC 1Vac
C_C1          N00174 N00832 382n
C_C2          N00832 N00194 382n
C_C3          N000680 N00832 815n
L_L1          0 N000680 2.29H
R_R1          N00269 N00174 1k
R_R2          0 N00194 1k
**** RESUMING "ac response.cir" ****
.END
ERROR -- Node N00832 is floating
```

Elliptic Filter

Add a dummy resistance of value $1\text{M}\Omega$ (1.0Meg) between the junction of $C1$, $C2$ and $C3$, and ground

Because this resistance is 1000 times the impedance of the filter it will not affect the response in any significant way

Simulate the ac response

Measure the maximum pass-band gain, the minimum pass-band gain and thus determine the pass-band ripple

Measure the maximum stop-band gain

Measure the cut-off (-3dB point) frequency

Elliptic Filter

It is convenient to have the output voltage displayed in dB

This can be done by using the Trace menu in PSpice A/D

Alternatively, and more conveniently, it is possible to place a VDB marker on the schematic

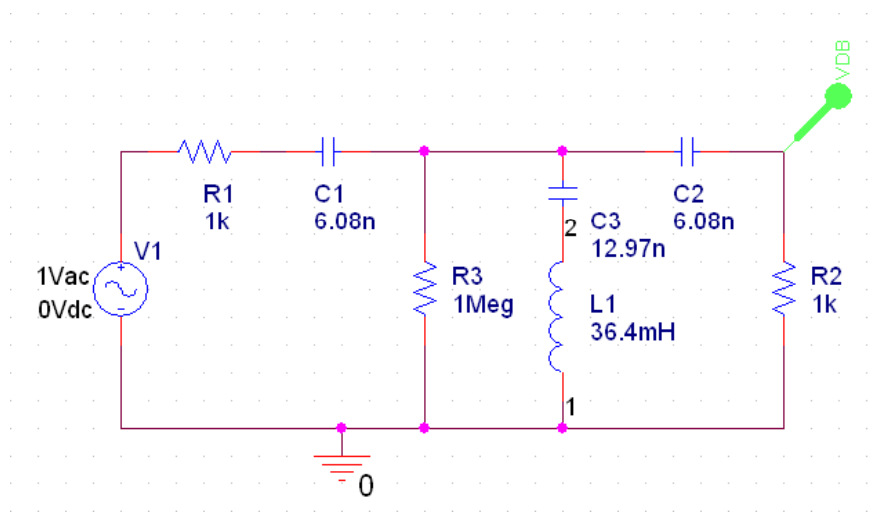
In Orcad Capture remove the voltage marker from the output node

Then select: PSpice / Markers / Advanced / db Magnitude of Voltage

Place VDB marker on output node

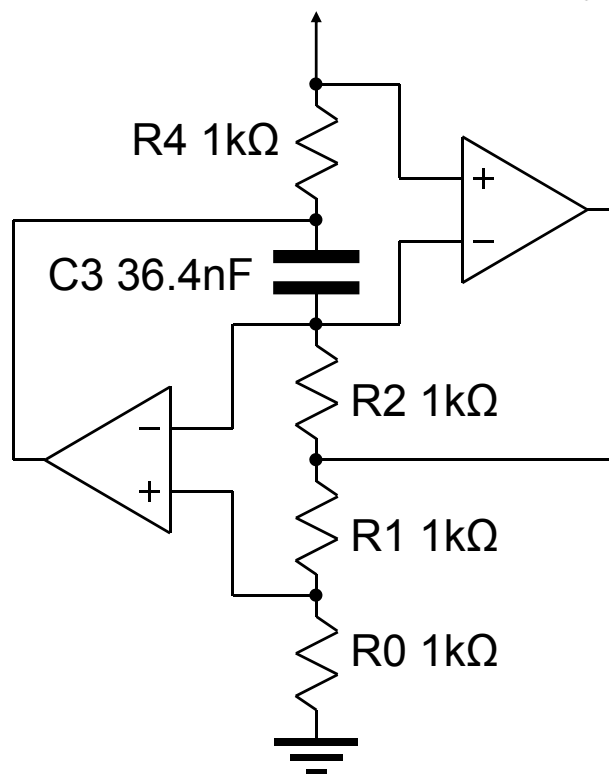
Elliptic Filter

Circuit schematic with VDB marker:

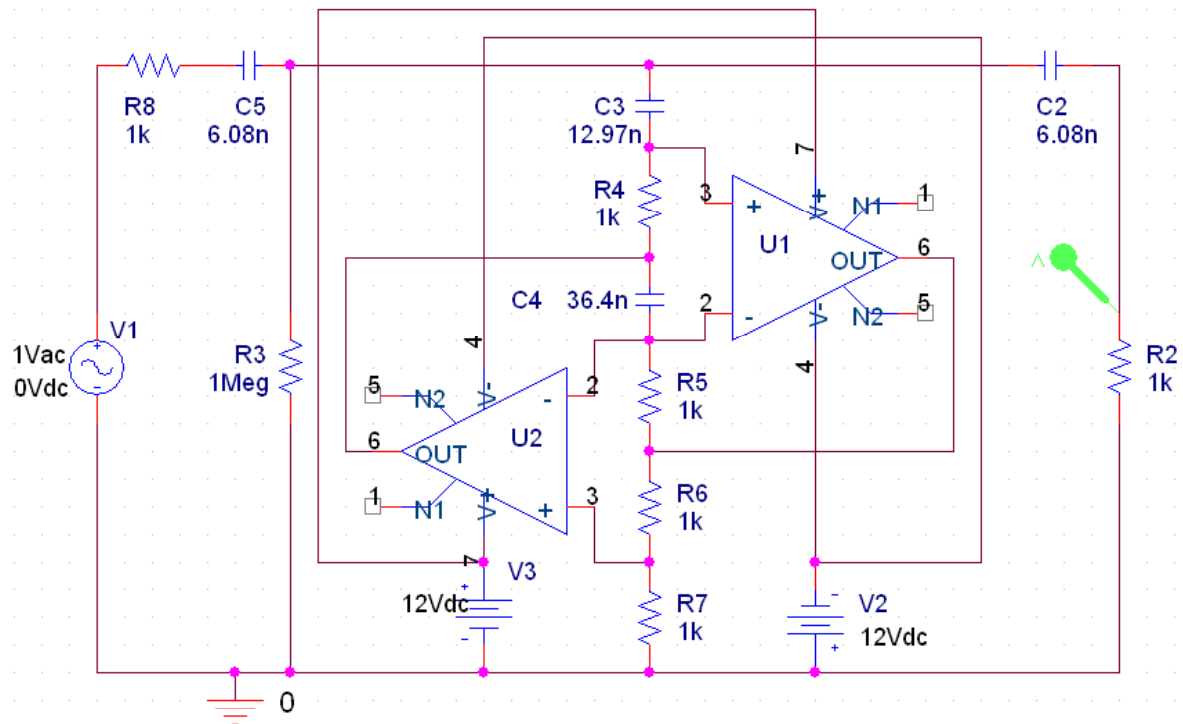


Elliptic Filter

Replace the grounded inductor by this simulated inductor using TL071 operational amplifiers:



Elliptic Filter



Elliptic Filter

Simulate the ac response of the active filter

Determine whether the response differs significantly from that of the passive filter

In particular consider the following properties of the response:

- Maximum pass-band gain
- Minimum pass-band gain
- Maximum stop-band gain
- Cut-off (-3dB point) frequency

What do you consider to be the relative advantages of the active and passive filters?

Session 4

Power Amplifiers Class C, B, A, AB

Power Amplifiers

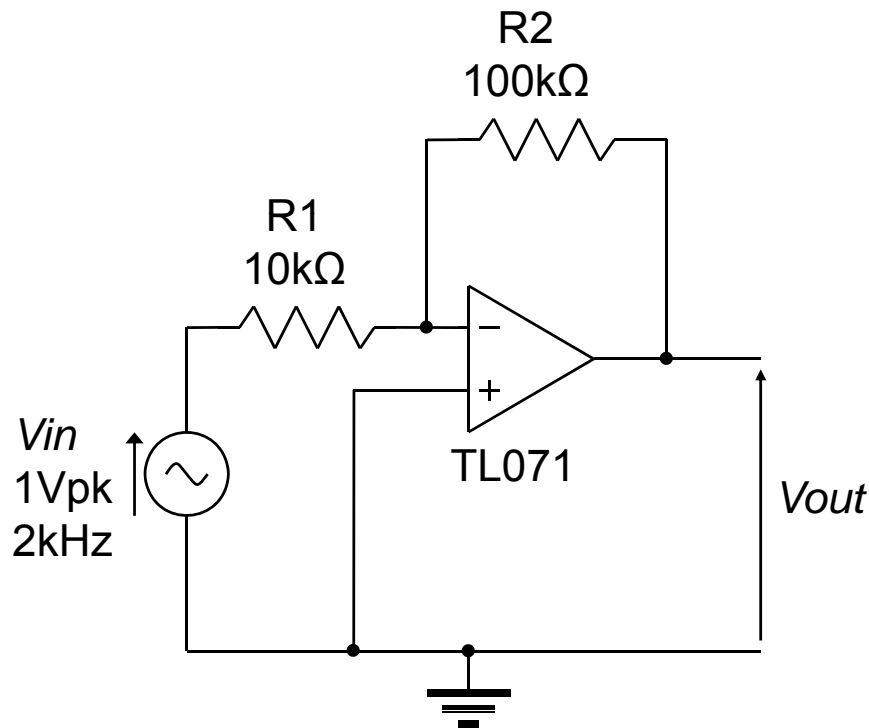
Operational amplifiers can provide gain with high linearity over the entire audio range (10Hz to 20kHz)

Loudspeakers in audio systems typically have an impedance of around 4Ω and require to be driven to at least $\pm 10V$

The power that operational amplifiers can generate into a load is limited by:

1. Voltage swing
2. Current limiting

Input the schematic of an inverting voltage amplifier using a TL071 operational amplifier and $\pm 15V$ power supplies



Power Amplifiers

Simulate the inverting amplifier with a 1V, 2kHz sinusoidal input over 1 ms with a maximum step size 0.1us

Confirm that the output waveform is sinusoidal, 10V in amplitude, and 180° out of phase with the input

Now connect a 4Ω load resistance between the output and ground

Notice that the output is no longer sinusoidal because of current limiting in the op-amp

By placing a current marker on the load resistance determine the positive and negative current limits of the op-amp

Power Amplifiers

Current limiting can be overcome by incorporating an emitter follower (common collector) stage on the output of the op-amp

Darlington transistors typically have a current gain of 500

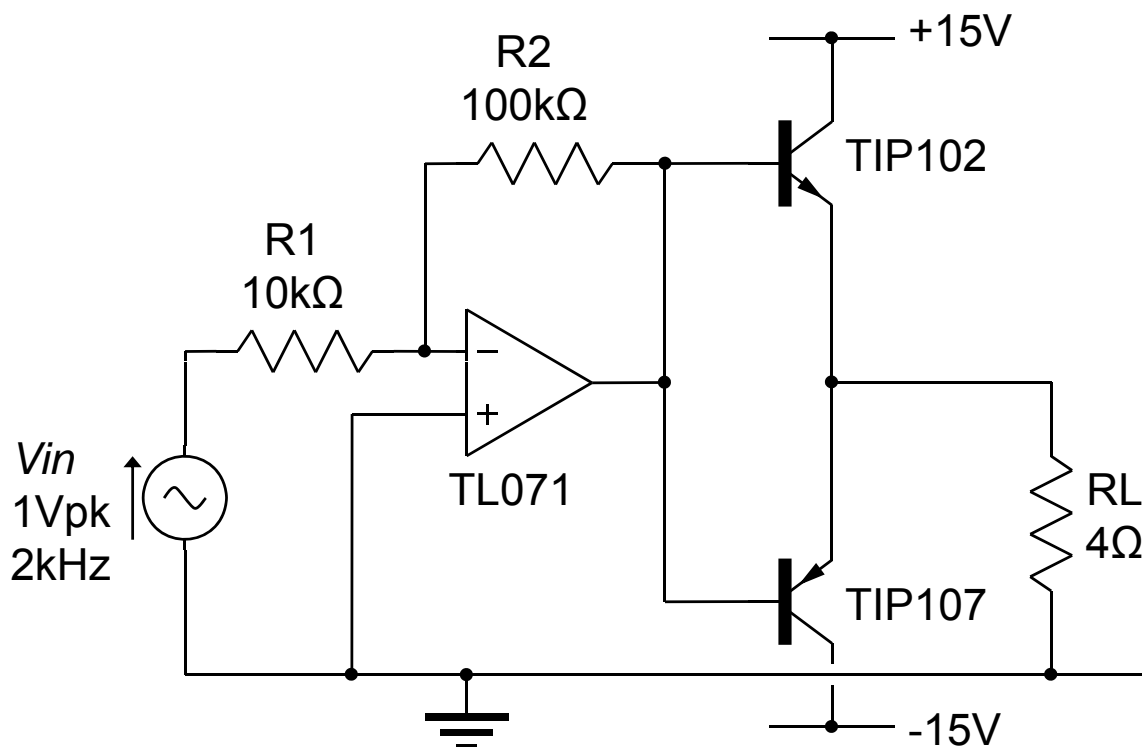
The peak current I_{\max} that will need to be supplied by the op-amp (for $\pm 10V$ output swing) is therefore:

$$I_{\max} = \frac{1}{500} \times \frac{10}{4} = 5\text{mA}$$

This is well within the capability of a TL071 op-amp

Modify your circuit to include an emitter-follower output stage (TIP102 and TIP107 are in the PSpice library: darlntn.olb)

Class C Power Amplifiers



Class C Power Amplifiers

Simulate the power amplifier with a 1V, 2kHz sinusoidal input

There now should be no limiting and the output voltage swing should have increased to around $\pm 10V$

Unfortunately the output stage has introduced cross-over distortion as the output goes from negative to positive

This occurs because Darlington transistors require a base-emitter voltage of 1.3V before they start to conduct

Zoom in on the distortion and note its characteristic shape

Class C Power Amplifiers

Power amplifiers are classified according to the proportion of a cycle that each of the output devices is conducting

This is a class C amplifier because each output device conducts for slightly less than one half cycle

Put current markers on the load resistor RL and on the emitters of the Darlington power transistors

Simulate the circuit and observe the current waveforms

Each Darlington power transistor should conduct for slightly less than one half cycle

Distortion in Power Amplifiers

The primary source of non-linear distortion is cross-over in the output stage

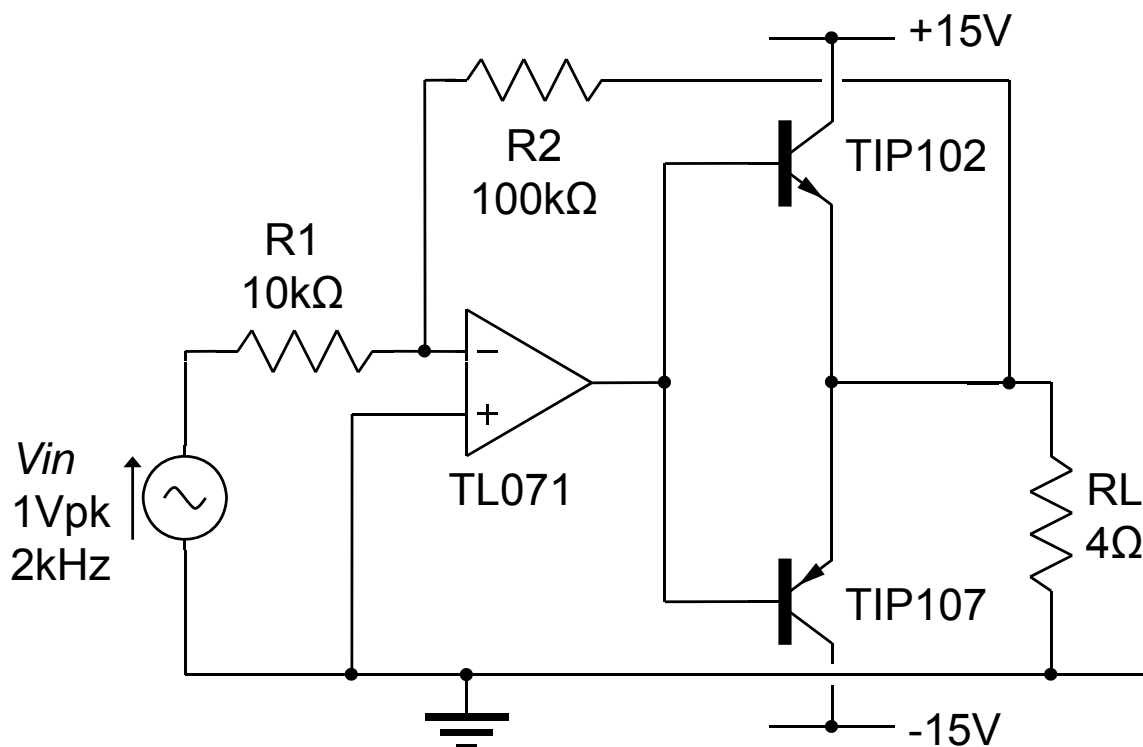
When feedback is taken from the output of the operational amplifier the loop does not include the output stage

The distortion can be reduced by extending the negative feedback loop to include the output stage

Modify your circuit diagram and again simulate the power amplifier with a 1V, 2kHz sinusoidal input

Zoom in on the distortion and note its characteristic shape

Distortion in Power Amplifiers



Distortion in Power Amplifiers

Fourier analysis allows the distorted output waveform to be resolved into fundamental and harmonics

The ratio of the harmonics to the fundamental is known as the harmonic distortion

To prepare for a Fourier analysis click on PSpice / Edit Simulation Profile and set:

Run to time = 10ms

Start saving data after = 5ms

Maximum step size = 0.1 μ s

This selects 10 cycles for analysis after 10 cycles (to allow any initial transients to decay)

Distortion in Power Amplifiers

Run the simulation - you should see 10 output cycles

Click: Trace / Fourier and: Plot / Axis Settings:

Set the X-axis to: User Defined / 0Hz to 24kHz / linear

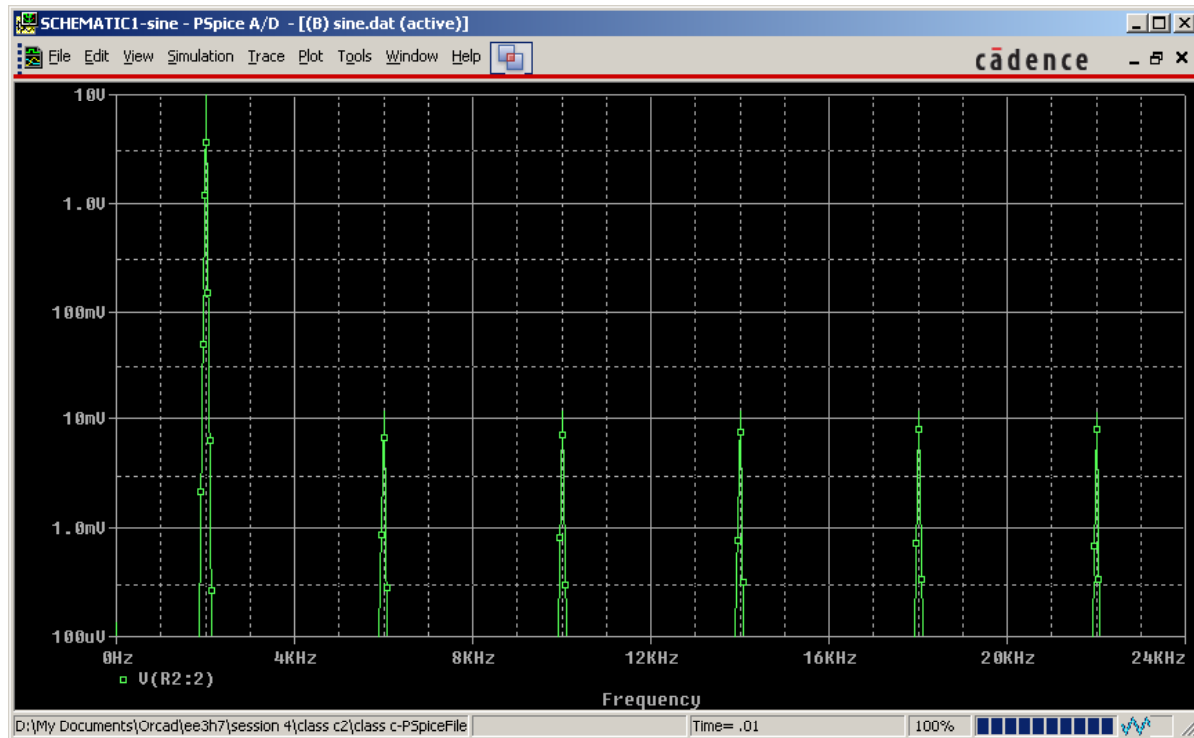
Set the Y-axis to: User Defined / 100 μ V to 10V / log

You will observe that only odd harmonics are present - this is because the cross-over distortion is an odd function


Now measure the size of the fundamental component and the odd harmonics

Calculate the third harmonic distortion as a percentage

Distortion in Power Amplifiers



Efficiency of Power Amplifiers

Place power markers:  on the load resistance and power supplies

Simulate the circuit using the previous simulation profile to obtain the instantaneous powers in the selected elements

Click on: Trace/Delete All Traces, then Trace/Add Trace

Click on: `AVGX(,)` from the rh pane, then: `W(RL)` from the lh pane, and type: `0.5ms <ret>`

You should now see a trace of the average power and: `AVGX(W(R5),0.5ms)` at the bottom of the window

Repeat for `W(V1)` and `W(V2)` ($\pm 15V$ power supplies)

Efficiency of Power Amplifiers

The power traces will vary over the first visible cycle (5.0ms to 5.5ms) and then become constant

The traces for the supplies are negative indicating that power flows out of these elements

Measure the average powers in the load and supplies at a point after 5.5ms

Thus calculate the efficiency:

$$\eta = \frac{\text{Average load power}}{\text{Average supply power}}$$


You should obtain a value of around 50%

Class B Power Amplifiers

The cross-over distortion in class-C amplifiers can be reduced by biasing the output transistors to the point of conduction

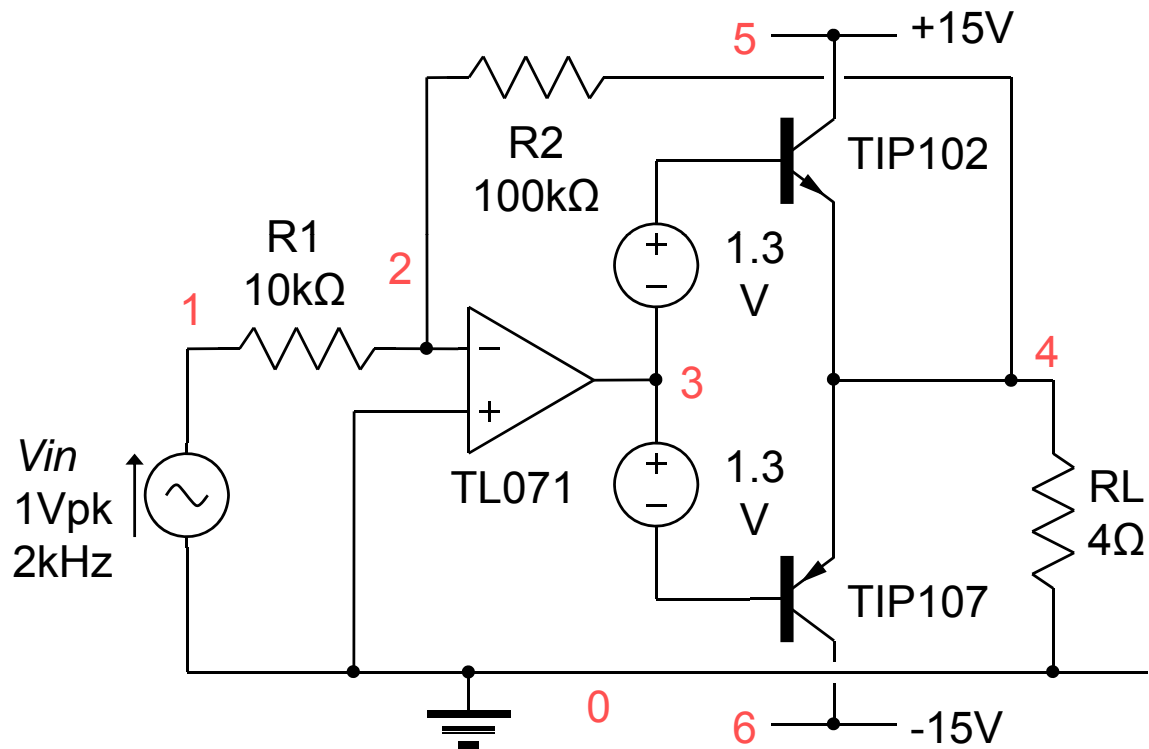
This is now becomes class B because each output device conducts for exactly one half cycle

Using SOURCE/VDC of 1.3 V for the bias voltages simulate the class-B power amplifier


Click on the  icon; this will display the bias point currents in the circuit

The bias current in the output darlington transistors should be around 12 mA

Class B Power Amplifiers



Class B Power Amplifiers

Click on the  icon; this will display the bias point powers - determine the quiescent power dissipation (from $\pm 15V$ supplies)

Inspect the output wave waveform - is there any visible cross-over distortion?

Inspect the current waveforms in the emitters of the output transistors and in the load - is this class B?

Measure the third harmonic distortion for this class-B amplifier

Measure the efficiency of this class-B amplifier

Class-B Power Amplifiers

Unfortunately class B amplifiers are thermally unstable

A rise in temperature of the output devices causes a reduction in the base-emitter voltage at which conduction occurs

Since the bias voltage remains constant the quiescent current in the output devices increases

As a result the output devices get hotter

This process is known as thermal runaway and makes class-B amplifiers unsatisfactory in practice

Class-A Power Amplifiers

Thermal runaway can be prevented by including extra resistances in the emitter circuits of the output devices

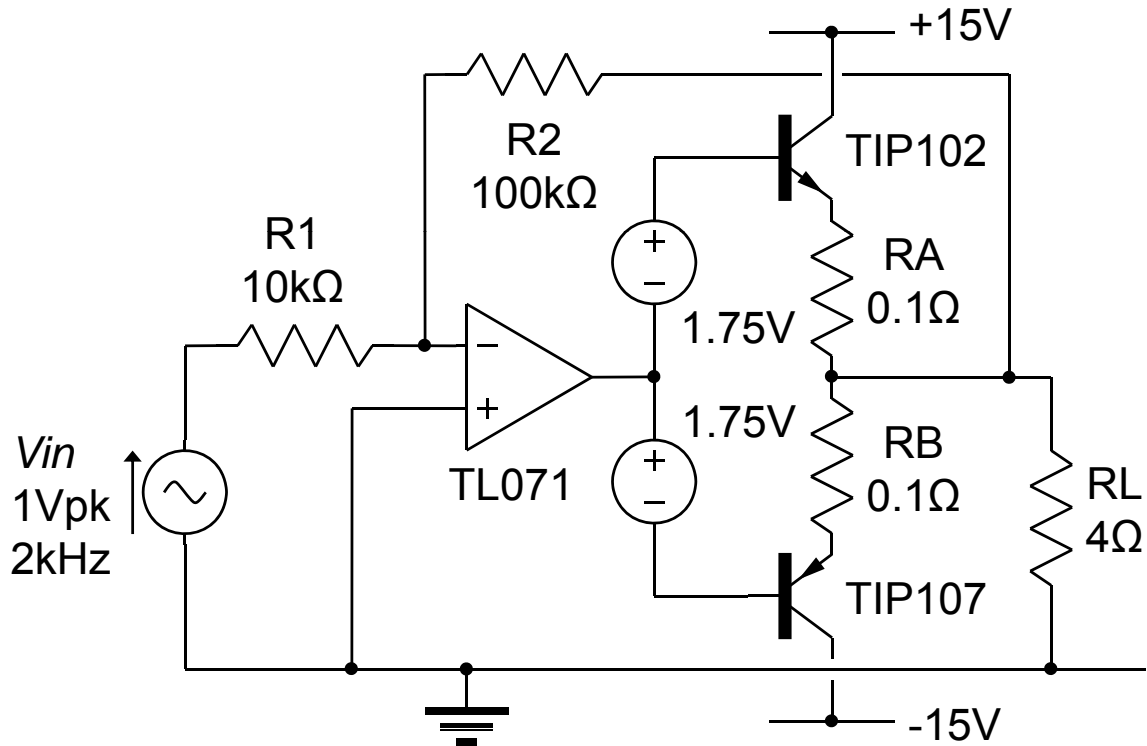
Now any increase in quiescent current reduces the base-emitter bias voltage thus stabilising the current

The bias voltages are increased to allow for the voltage drop across the emitter resistances

If the quiescent current is set to half the peak load current then both output devices conduct for the complete cycle

This is known as class-A operation

Class-A Power Amplifiers



Class-A Power Amplifiers

Peak load current $I_{L_{peak}} = 15V/4\Omega = 3.75A$

Ideal quiescent current $I_q = 3.75/2 = 1.875A$

Simulate the class-A power amplifier

Measure the following:

quiescent current in output transistors

quiescent power dissipation

fraction of the cycle for which the output devices conduct

third harmonic distortion

efficiency

Compare the quiescent power dissipation with the average power into the load

Class-AB Power Amplifiers

In most cases the very high power dissipation of class-A amplifiers is unacceptable

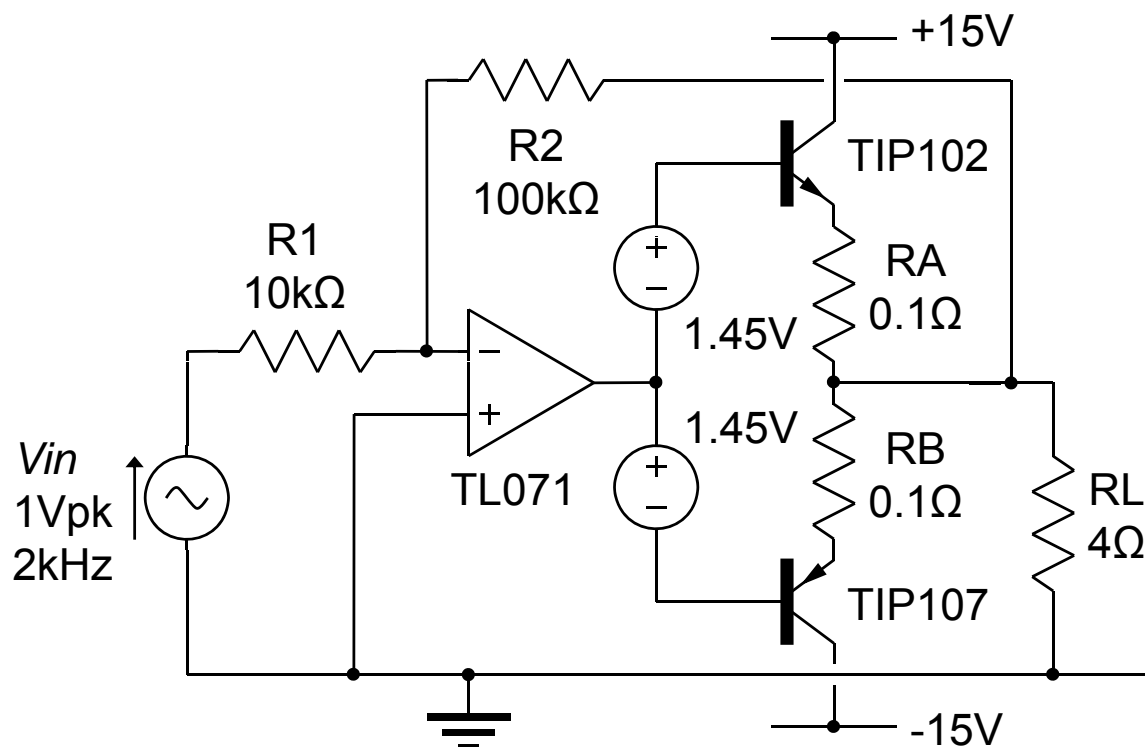
As the name implies Class-AB is intended to provide the linearity of class-A with the efficiency of class-B

The quiescent current on the output devices is set to a small fraction of the peak load current

The output devices conduct for slightly more than half a cycle

In practice the distortion in class-AB amplifiers is worse than class-A and the efficiency less than class-B

Class-AB Power Amplifiers



Class-AB Power Amplifiers

The circuit is the same as that of the class-A amplifier but the bias voltages have been reduced.

Simulate the class-AB power amplifier

Measure the following:

- quiescent current in output transistors

- quiescent power dissipation

- fraction of the cycle for which the output devices conduct

- third harmonic distortion

- efficiency

Compare the quiescent power dissipation with the average power into the load

Analogue Simulation

Session 5

DC-DC Converters
Buck Converters

DC-DC Converters

Dc-dc converters are commonly used in regulated power supplies and in motor drives

The input is usually an unregulated dc voltage derived from mains power

In this course we shall examine five converter topologies:

- Step-down (buck) converter
- Step-up (boost) converter
- Step-up/down (buck-boost) converter
- Flyback converter
- Forward converter

Output voltage/current control is achieved by varying the duty cycle of the switch

Buck Converters

Buck converters produce a load (output) voltage v_o that is lower than the input voltage v_s

Normally buck converters operate at a fixed frequency $f_s=1/T_s$ with a variable duty cycle k

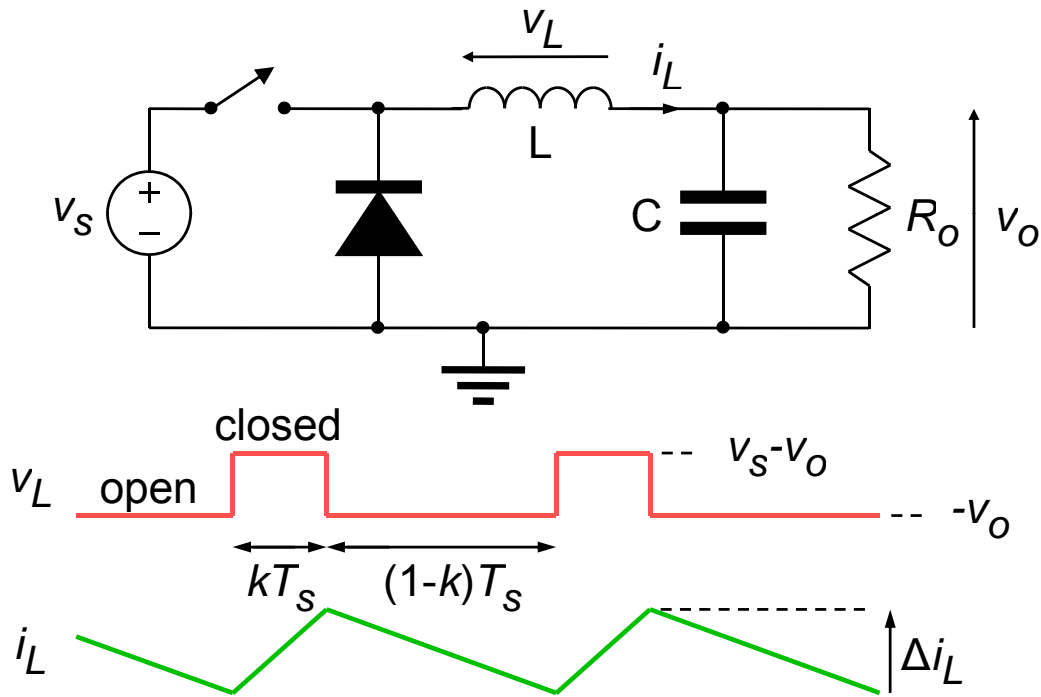
The switch is a semiconductor device - usually a bipolar junction transistor (BJT) or a MOSFET

A fast-recovery flywheel diode is also required

Buck converters can operate in one of two modes:

- Continuous-conduction mode
- Discontinuous-conduction mode

Buck Converters



Buck Converters

Continuous conduction mode:

$$L \frac{di_L}{dt} = -v_o$$

$$L \frac{di_L}{dt} = v_s - v_o$$

$$\Delta i_L = -(1-k)T_s \frac{-v_o}{L}$$

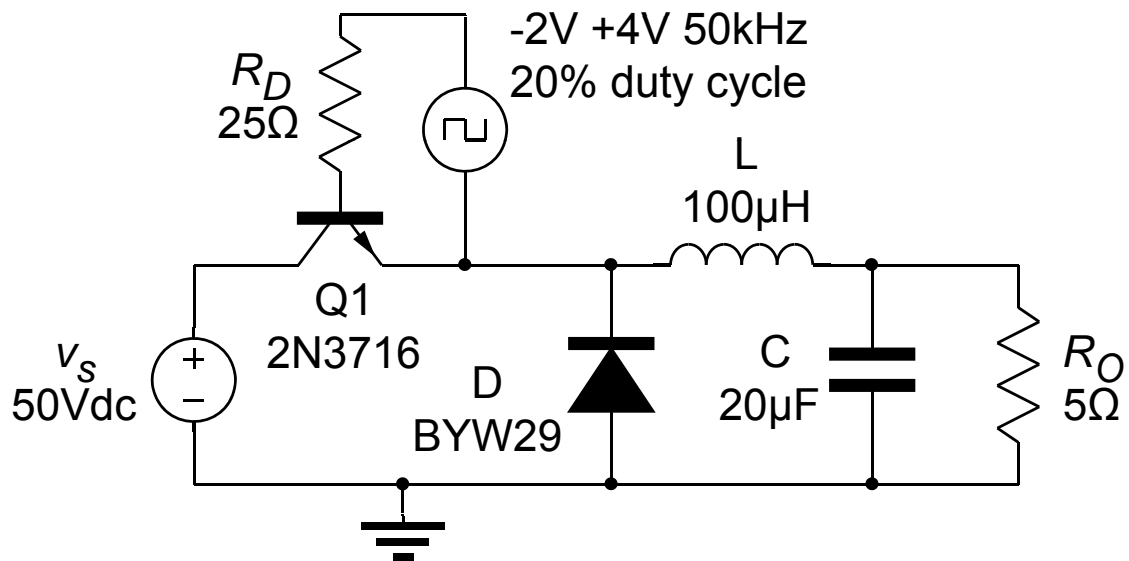
$$\Delta i_L = kT_s \frac{v_s - v_o}{L}$$

$$-(1-k)T_s \frac{-v_o}{L} = kT_s \frac{v_s - v_o}{L}$$

$$(1-k)v_o = k(v_s - v_o)$$

$$v_o = kv_s$$

Buck Converters



Buck Converters

For the base driver of the transistor select from SOURCE:
VPULSE and set the parameters:

$$V1 = -2.0$$

$$V2 = 4.0$$

$$TD = 0.0$$

$$TR = 10.0n$$

$$TF = 10.0n$$

$$PW = 4.0u$$

$$PER = 20.0u$$

Place voltage markers on the transistor emitter and output,
and current markers on the inductor and transistor collector

Buck Converters

Click on: PSpice / Edit Simulation Profile and set:

Run to time = 2040 μ s

Start saving data after = 2000 μ s

Maximum step size = 0.1 μ s

This selects 2 cycles for analysis after 100 cycles (to allow any initial transients to decay)

Simulate the buck converter

Measure the output voltage (after it has settled down) and compare it with the theoretical value

Buck Converters

Measure the conduction voltage of the diode

Place power markers on the input voltage supply and the load resistance

Plot the average of these quantities:

$AVGX(W(V1),20u)$

$AVGX(W(RL),20u)$

Hence calculate the efficiency of the buck converter

Buck Converters

Replace the bipolar transistor by a MOSFET

Change the VPULSE parameters to:

$$V1 = 0.0$$

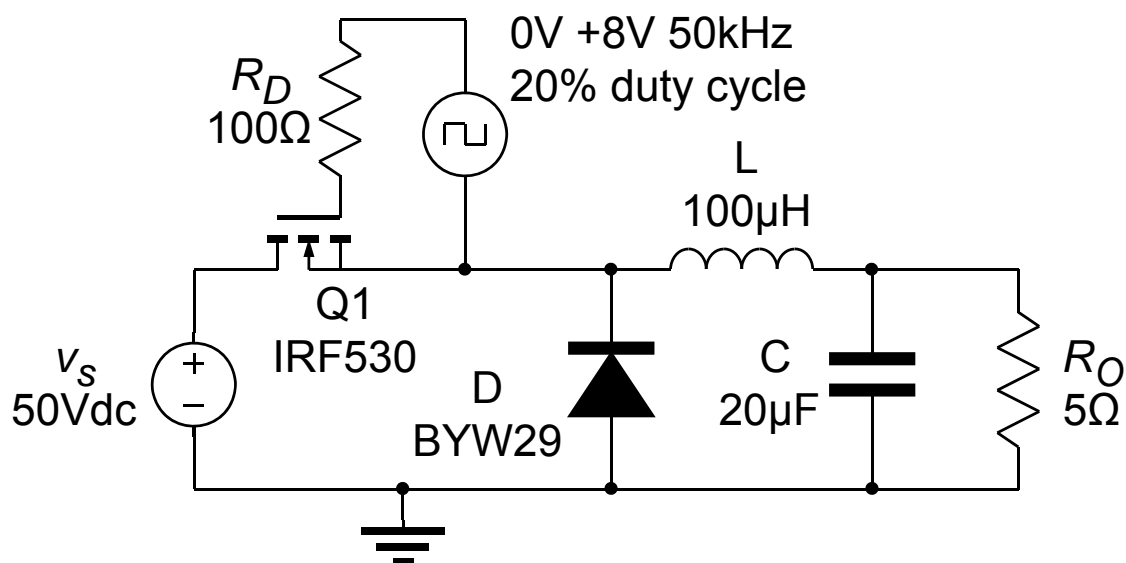
$$V2 = 8.0$$

Set the maximum step size to $0.1\mu\text{s}$ and simulate the buck converter over the time range $0\text{-}2040\mu\text{s}$

Measure the output voltage and compare it with the theoretical value

Measure the efficiency and compare it with that of the converter using a bipolar transistor switch

Buck Converters



Buck Converters

The regulation N of a converter is a measure of the change in output voltage v_o in response to a change in output current i_o

$$N = \frac{dv_o}{di_o} \approx \frac{\Delta v_o}{\Delta i_o}$$

Regulation is identical to output resistance

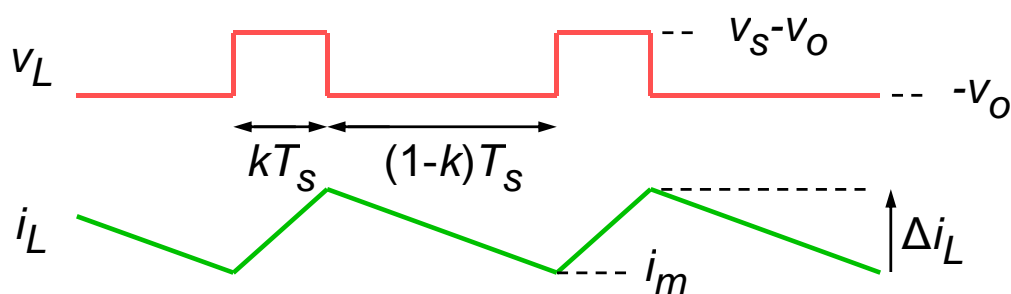
Measure the dc output of the buck converter

Then change the load resistance R_o to 4Ω and again measure the output voltage of the converter

Hence determine the regulation N of the buck converter

Buck Converters

Continuous/Discontinuous Boundary



Continuous conduction will occur when the average load current i_o is such that i_m is greater than 0

$$i_o = i_m + \frac{\Delta i_L}{2}$$

$$i_o \geq \frac{\Delta i_L}{2} = \frac{kT_s}{2L} (v_s - v_o)$$

Buck Converters

Continuous/Discontinuous Boundary

With the present design of buck converter discontinuous conduction will occur when:

$$i_o < \frac{0.2 \times 20 \times 10^{-6}}{2 \times 100 \times 10^{-6}} (50 - 10) = 0.8 \text{ A}$$

This current corresponds to a load resistance R_o of 12.5Ω

Increase the load resistance to 20Ω and simulate the buck converter over the time range 0-2040 μ s

Inspect the inductor current waveform

Buck Converters

Continuous/Discontinuous Boundary

Measure the output voltage of the converter

Change the load resistance R_o to 15Ω and again measure the output voltage

Hence determine the regulation M in discontinuous mode

Session 6

DC-DC Converters Boost Converters Buck-Boost Converters

Boost Converters

Boost converters produce a load (output) voltage v_o that is greater than the input voltage v_s

Normally boost converters operate at a fixed frequency f_s with a variable duty cycle (mark-space ratio) k

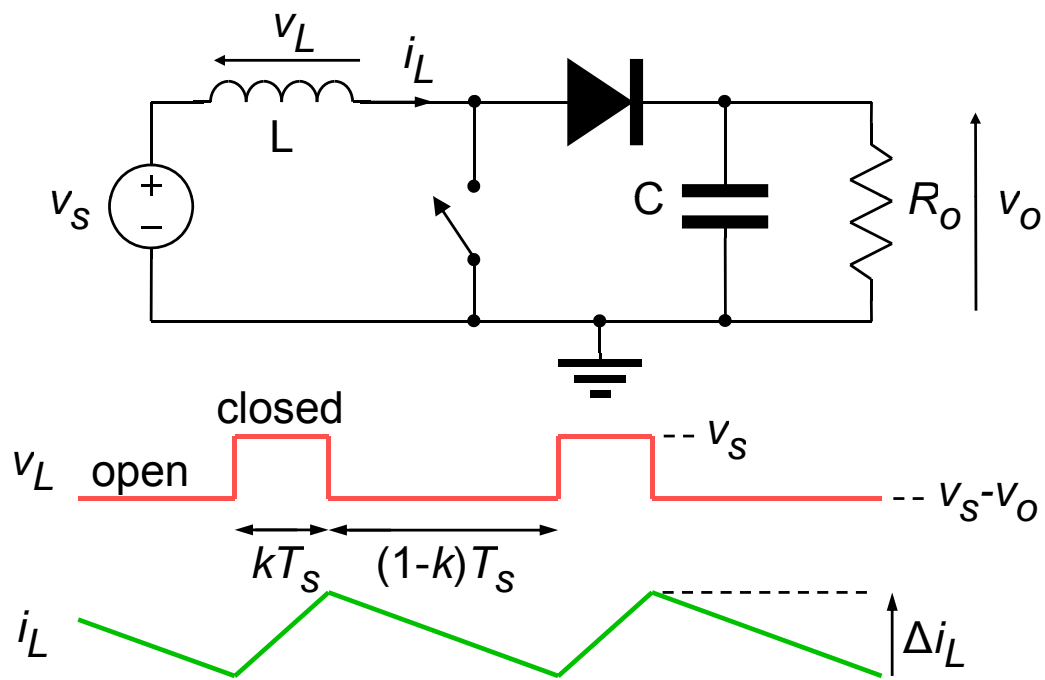
Boost converters can operate in one of two modes:

Continuous-conduction mode

Discontinuous-conduction mode

As with buck converters, the behaviour is quite different in these two modes

Boost Converters



Boost Converters

Continuous conduction mode:

$$L \frac{di_L}{dt} = v_s$$

$$L \frac{di_L}{dt} = v_s - v_o$$

$$\Delta i_L = kT_s \frac{v_s}{L}$$

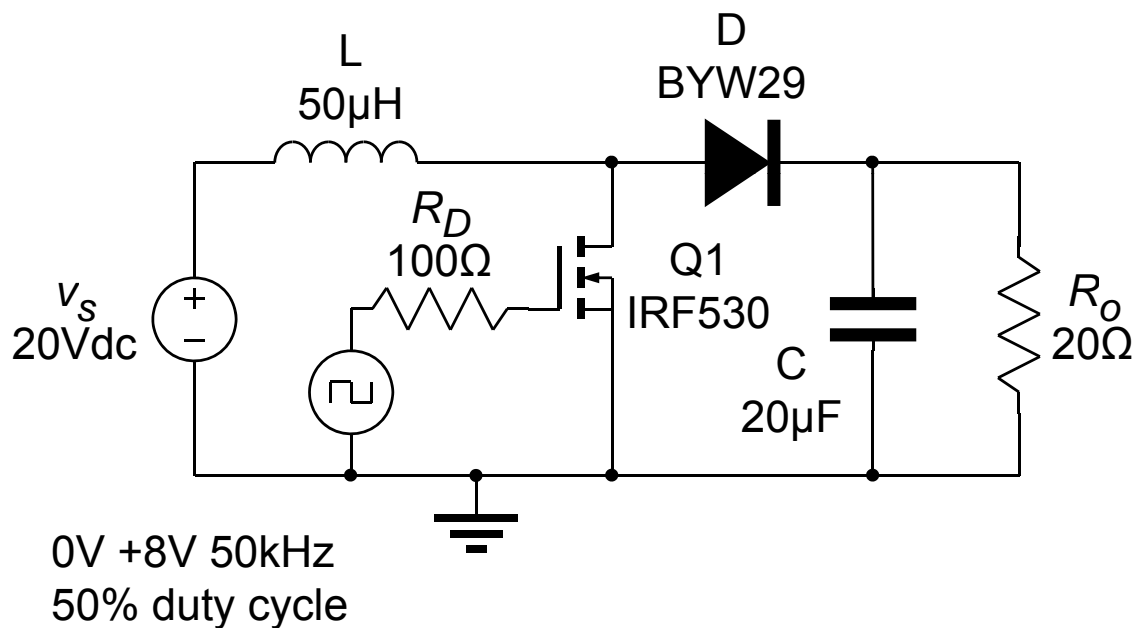
$$\Delta i_L = -(1-k)T_s \frac{v_s - v_o}{L}$$

$$kT_s \frac{v_s}{L} = -(1-k)T_s \frac{v_s - v_o}{L}$$

$$kv_s = (1-k)(v_o - v_s)$$

$$v_o = \frac{v_s}{(1-k)}$$

Boost Converters



Boost Converters

For the gate driver of the MOSFET select from SOURCE:
VPULSE and set the parameters:

$$V1 = 0.0$$

$$V2 = 8.0$$

$$TD = 0.0$$

$$TR = 10.0n$$

$$TF = 10.0n$$

$$PW = 10.0u$$

$$PER = 20.0u$$

Place voltage markers on the MOSFET drain and output,
and current markers on the inductor and MOSFET drain

Click on PSpice/Edit Simulation Profile and set:

Run to time = 2040 μ s

Start saving data after = 2000 μ s

Maximum step size = 0.1 μ s

Measure the maximum and minimum output voltages

Calculate the average output voltage and compare it with the theoretical value

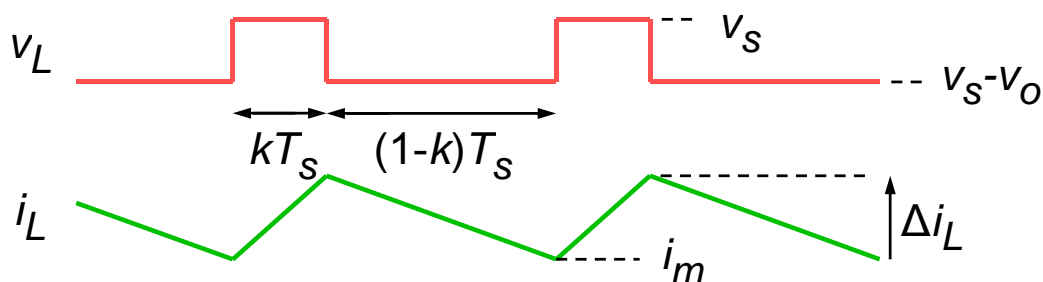
Calculate the output ripple of the boost converter

Determine the efficiency of the boost converter

By changing the load resistance R_o to 15Ω determine the regulation of the boost converter

Boost Converters

Continuous/Discontinuous Boundary



Continuous conduction will occur when the average load current i_o is such that i_m is greater than 0

$$i_o = (1-k) \left(i_m + \frac{\Delta i_L}{2} \right)$$

$$i_o \geq (1-k) \frac{\Delta i_L}{2} = \frac{k(1-k)T_s}{2L} V_s$$

Boost Converters

Continuous/Discontinuous Boundary

With the present design of buck converter discontinuous conduction will occur when:

$$i_o \geq \frac{0.5(1-0.5) \times 20 \times 10^{-6}}{2 \times 50 \times 10^{-6}} 20 = 1.0 \text{ A}$$

This current corresponds to a load resistance of 40.0Ω

Increase the load resistance R_o to 60Ω and simulate the boost converter

Measure the output voltage and inspect the inductor current waveform

Buck-Boost Converters

Buck-boost converters produce a load (output) voltage v_o that can be less or greater than the input voltage v_s

Normally buck-boost converters operate at a fixed frequency f_s with a variable duty cycle (mark-space ratio) k

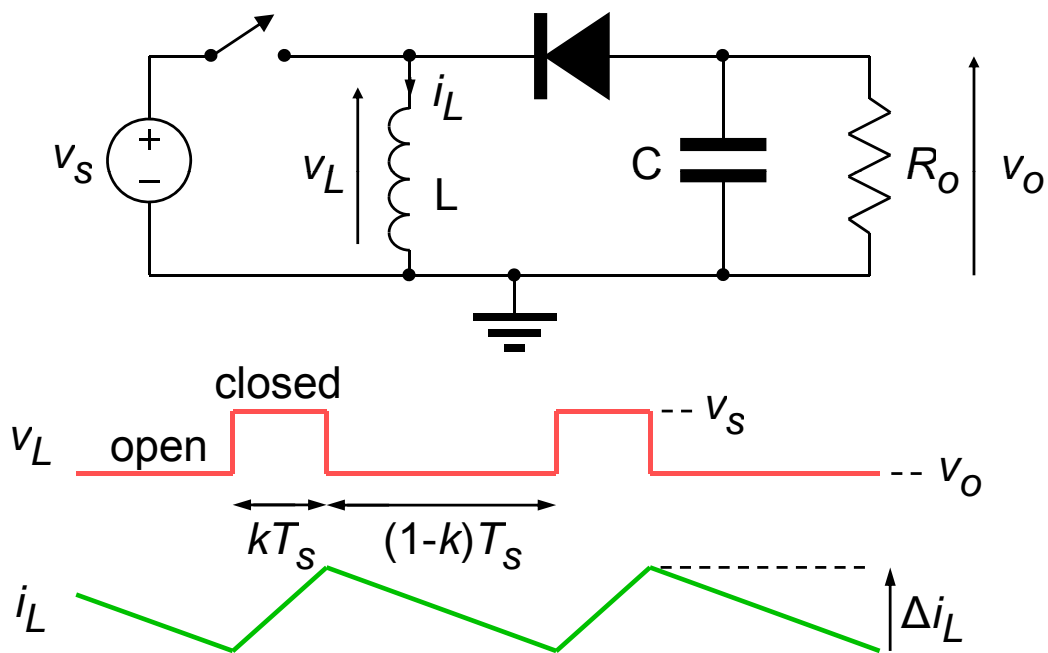
Buck-boost converters invert the voltage so that a positive input voltage v_s gives a negative load (output) voltage v_o

Boost converters can operate in one of two modes:

Continuous-conduction mode

Discontinuous-conduction mode

Buck-Boost Converters



Buck-Boost Converters

Continuous conduction mode:

$$L \frac{di_L}{dt} = v_s$$

$$L \frac{di_L}{dt} = v_o$$

$$\Delta i_L = kT_s \frac{v_s}{L}$$

$$\Delta i_L = -(1-k)T_s \frac{v_o}{L}$$

$$kT_s \frac{v_s}{L} = -(1-k)T_s \frac{v_o}{L}$$

$$(k-1)v_o = kv_s$$

$$v_o = \frac{k}{(k-1)} v_s$$

Buck-Boost Converters

For a 50% duty cycle:

$$v_o = \frac{k}{(k-1)} v_s = \frac{0.5}{-0.5} v_s = -v_s$$

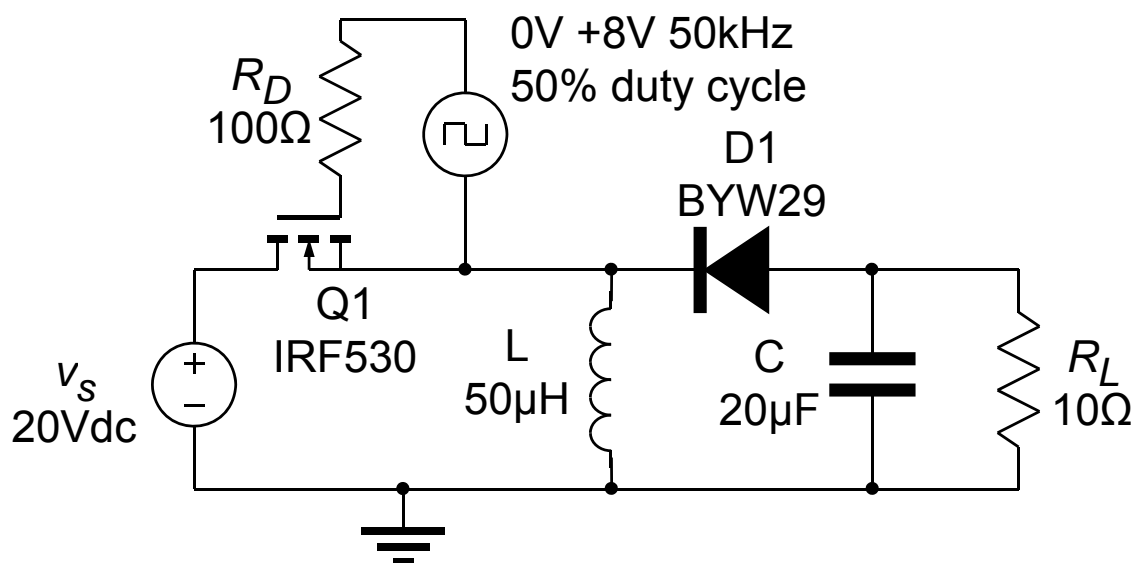
For a 20% duty cycle:

$$v_o = \frac{k}{(k-1)} v_s = \frac{0.2}{-0.8} v_s = -0.25v_s$$

For an 80% duty cycle:

$$v_o = \frac{k}{(k-1)} v_s = \frac{0.8}{-0.2} v_s = -4v_s$$

Buck-Boost Converters



Buck-Boost Converters

For the gate driver of the MOSFET select from SOURCE: VPULSE and set the parameters:

V1 = 0.0	V2 = 8.0	
TD = 0.0	TR = 10.0n	TF = 10.0n
PW = 10.0u	PER = 20.0u	

Place voltage markers on the MOSFET source and output, and current markers on the inductor and MOSFET drain

Click on PSpice/Edit Simulation Profile and set:

Run to time = 2040 μ s
Start saving data after = 2000 μ s
Maximum step size = 0.1 μ s

Buck-Boost Converters

Measure the maximum and minimum output voltages

Calculate the average output voltage and compare it with the theoretical value

Calculate the output ripple of the buck-boost converter

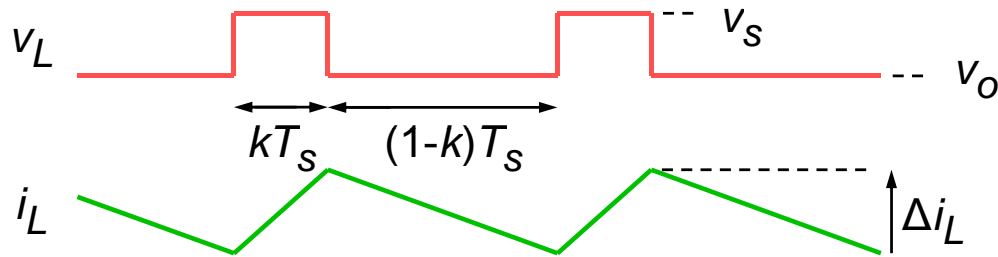
Determine the efficiency of the buck-boost converter

By changing the load resistance R_o to 15 Ω determine the regulation of the buck-boost converter

Change the duty cycle to 40% and to 60%, and measure the output voltage - compare with theoretical values

Buck-Boost Converters

Continuous/Discontinuous Boundary



Continuous conduction will occur when the load current i_o is such that i_m is greater than 0

$$i_o = (1-k) \left(i_m + \frac{\Delta i_L}{2} \right)$$

$$i_o \geq (1-k) \frac{\Delta i_L}{2} = \frac{(1-k)^2 T_s}{2L} v_o$$

Buck-Boost Converters

Continuous/Discontinuous Boundary

With the present design of buck converter discontinuous conduction will occur when:

$$i_o \geq \frac{0.5^2 \times 20 \times 10^{-6}}{2 \times 50 \times 10^{-6}} 20 = 1.0 \text{ A}$$

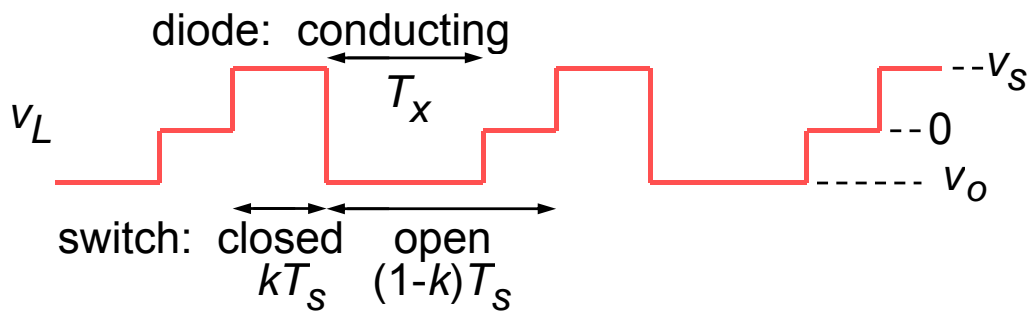
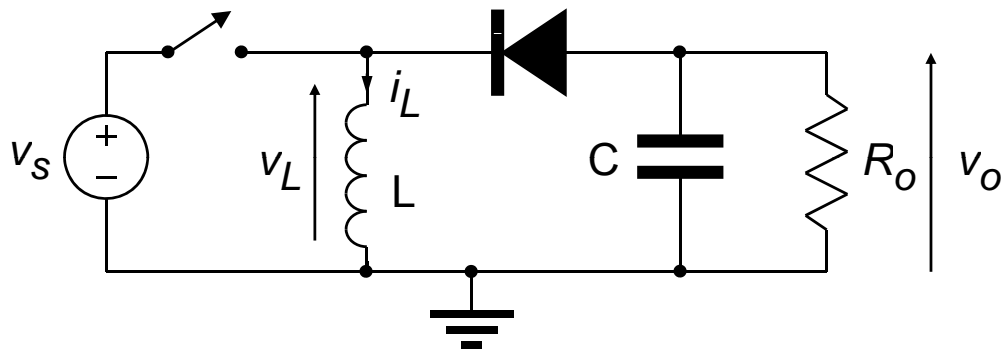
This current corresponds to a load resistance of 20Ω

Increase the load resistance R_o to 40Ω and simulate the buck-boost converter with duty cycle 50%

Inspect the inductor current waveform

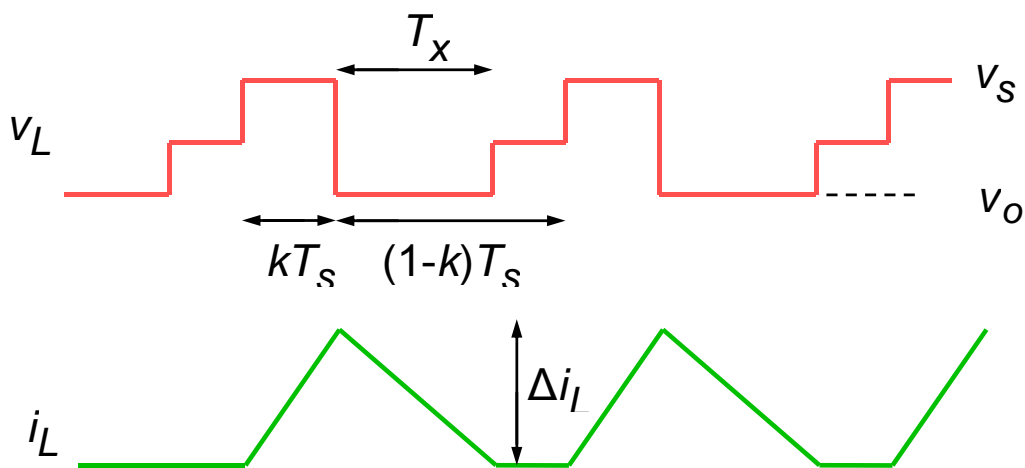
Buck-Boost Converters

Discontinuous mode



Buck-Boost Converters

Discontinuous mode



$$\Delta i_L = kT_s \frac{v_s}{L_1}$$

$$\Delta i_L = -T_x \frac{v_o}{L_1}$$

Buck-Boost Converters

Discontinuous mode

During period that switch is closed the current in the inductor increases, storing energy

When the switch is opened this energy is transferred to the capacitor and is dissipated in the load

The converter therefore operates at constant power

Energy stored in inductor:

$$E = \frac{1}{2} L \Delta i_L^2 = \frac{1}{2} L \left(\frac{k T_s V_s}{L} \right)^2 = \frac{1}{2L} k^2 T_s^2 V_s^2$$

Power:

$$P = f_s E = \frac{1}{2L} k^2 T_s V_s^2$$

Buck-Boost Converters

Discontinuous mode

For the circuit that you are simulating:

$$\begin{aligned} P &= \frac{V_o^2}{R_o} = \frac{1}{2L} k^2 T_s V_s^2 \\ &= \frac{0.5^2 \times 20 \times 10^{-6} \times 20^2}{2 \times 50 \times 10^{-6}} \\ &= 20 \text{ W} \end{aligned}$$

$$\begin{aligned} P &= \frac{V_o^2}{R_o} = 20 \text{ W} \\ V_o &= \sqrt{P \times R_o} = \sqrt{20 \times 40} \\ &= -28.28 \text{ V} \end{aligned}$$

Measure the dc output voltage of the buck-boost converter and compare it with this theoretical value

Session 7

DC-DC Converters Flyback Converters

Flyback Converters

Flyback converters are similar to buck-boost converters but incorporate a transformer between input and output

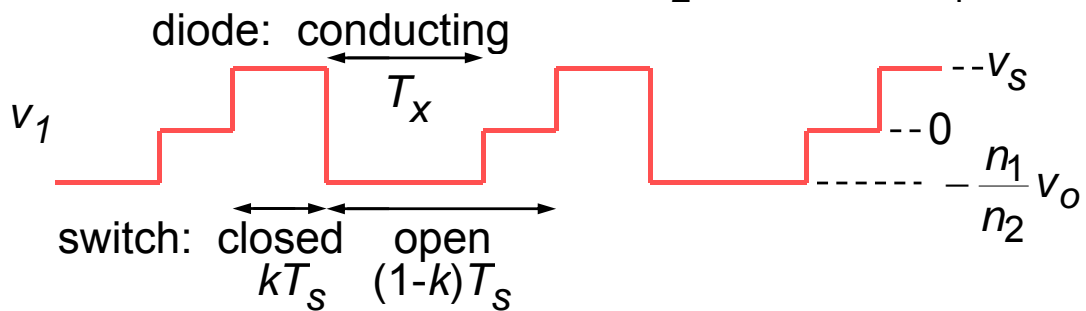
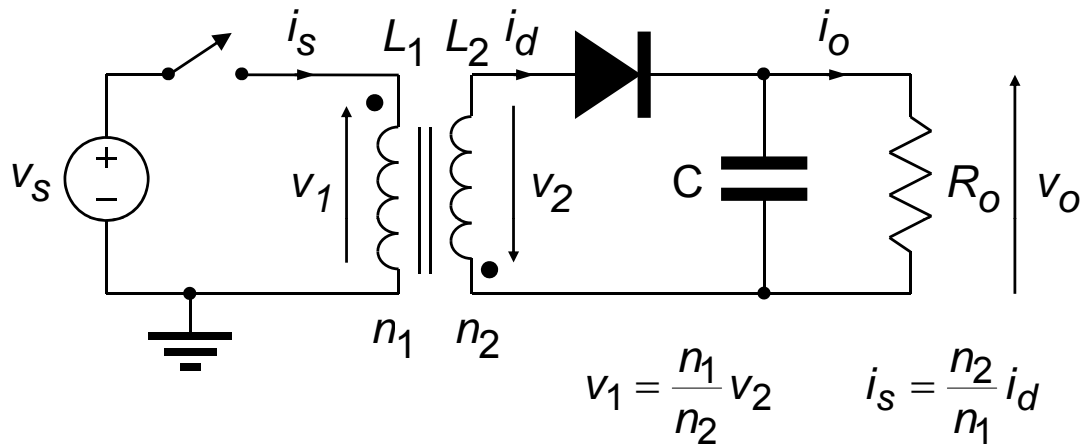
Flyback converters have two advantages over buck-boost:

- There is electrical isolation between input and output
- A wider ratio between input and output voltages is possible

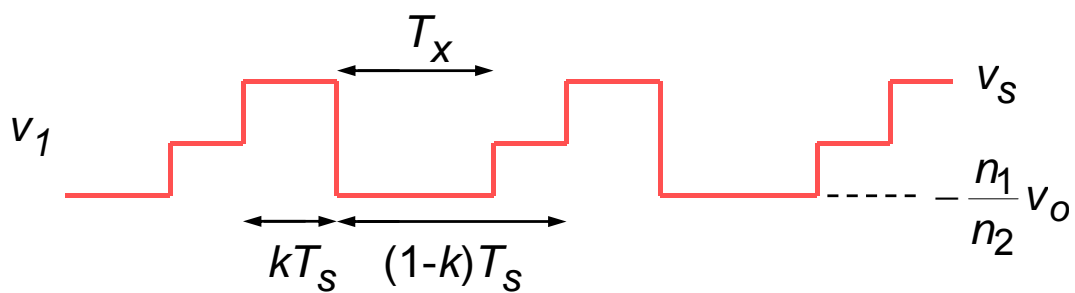
Flyback converters produce a load (output) voltage v_o that can be less or greater than the input voltage v_s

Flyback converters are used in discontinuous mode - the current in the transformer primary drops to zero each cycle

Flyback Converters



Flyback Converters - Primary

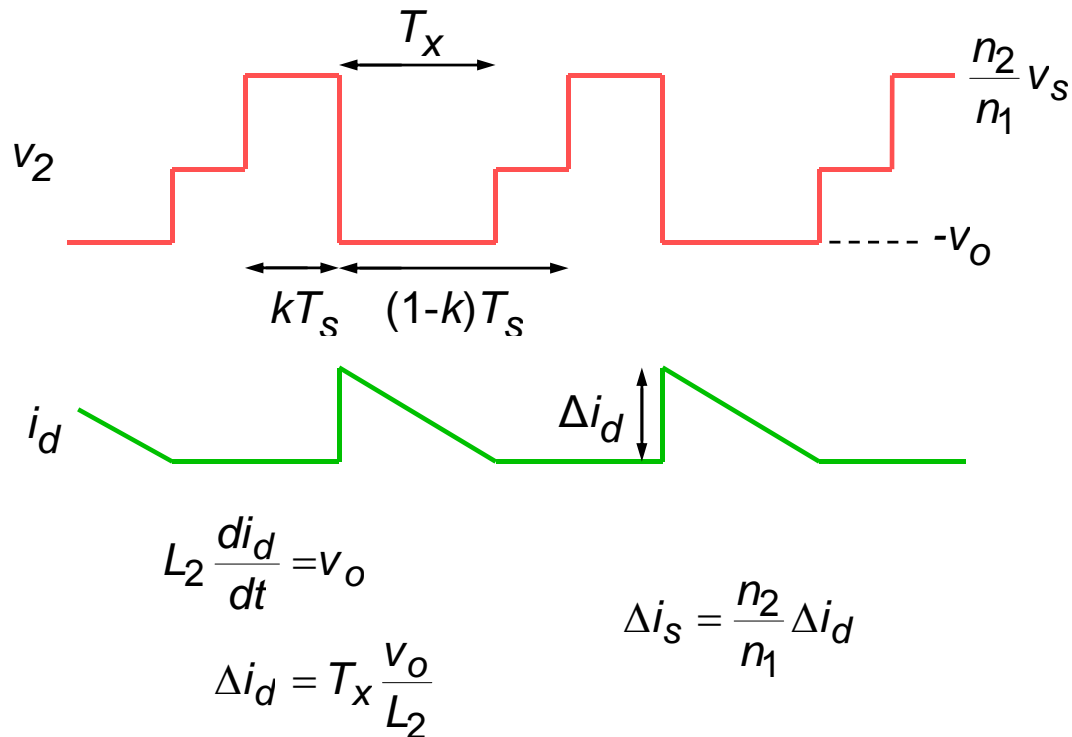


$$L_1 \frac{di_s}{dt} = v_s$$

$$\Delta i_s = kT_s \frac{v_s}{L_1}$$

$$\Delta i_s = \frac{n_2}{n_1} \Delta i_d$$

Flyback Converters - Secondary



Flyback Converters

The inductance L of an inductor is proportional to the square of the number of turns n :

$$L_1 = kn_1^2 \quad L_2 = kn_2^2 = L_1 \frac{n_2^2}{n_1^2}$$

So that:

$$\Delta i_s = kT_s \frac{v_s}{L_1} \quad \Delta i_s = \frac{n_2}{n_1} \Delta i_d \quad \Delta i_d = T_x \frac{v_o}{L_2}$$

$$kT_s \frac{v_s}{L_1} = \frac{n_2}{n_1} T_x \frac{v_o}{L_2}$$

$$T_x = kT_s \frac{v_s}{L_1} \frac{n_1}{n_2} \frac{L_2}{v_o} = kT_s \frac{n_2}{n_1} \frac{v_s}{v_o}$$

Flyback Converters

For discontinuous conduction mode

$$T_x < (1-k)T_s$$

So that:

$$T_x = kT_s \frac{n_2 V_s}{n_1 V_o} < (1-k)T_s$$

$$\frac{n_1}{n_2} \times \frac{V_o}{V_s} > \frac{k}{1-k}$$

For a 50% duty cycle ($k=0.5$):

$$\frac{n_1}{n_2} \times \frac{V_o}{V_s} > \frac{k}{1-k} = 1$$

$$\frac{n_2}{n_1} < \frac{V_o}{V_s}$$

Flyback Converters

During period that switch is closed the current in the primary winding increases, storing energy

When the switch is opened this energy is transferred to the secondary winding and is dissipated in the load

Flyback converters therefore operate at constant power

Energy stored in inductor:

$$E = \frac{1}{2} L_1 \Delta i_s^2 = \frac{1}{2} L_1 \left(\frac{k T_s V_s}{L_1} \right)^2 = \frac{1}{2 L_1} k^2 T_s^2 V_s^2$$

Power:

$$P = f_s E = \frac{1}{2 L_1} k^2 T_s V_s^2$$

Flyback Converters

Example: a 5kHz flyback converter with a 6V input generating a 400V output into a 20kΩ load

$$\text{For } k = 0.5: \quad \frac{n_2}{n_1} < \frac{400}{6} = 66.7 \quad \text{so let: } \frac{n_2}{n_1} = 20$$

$$\text{Power:} \quad P = \frac{v_o^2}{R_o} = \frac{400^2}{20000} = 8\text{W}$$

$$\text{So:} \quad \frac{1}{2L_1} k^2 T_s v_s^2 = 8$$

$$L_1 = \frac{1}{16} k^2 T_s v_s^2 = \frac{1}{16} \times 0.5^2 \times 200 \times 10^{-6} \times 6^2 = 112.5\mu\text{H}$$

Flyback Converters

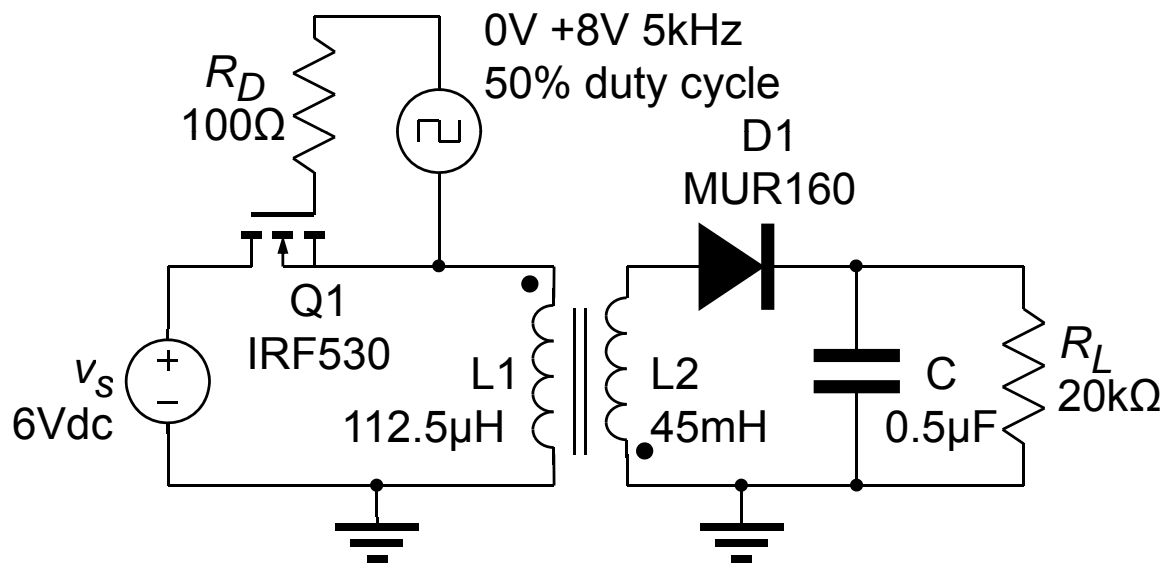
The peak current in the primary winding is:

$$\begin{aligned} \Delta i_s &= k T_s \frac{v_s}{L_1} \\ &= 0.5 \times 200 \times 10^{-6} \frac{6}{112.5 \times 10^{-6}} \\ &= 5.33 \text{ A} \end{aligned}$$

The primary inductance $L_1 = 112.5\mu\text{H}$ and the turns ratio is $n_2/n_1 = 20$:

$$\begin{aligned} L_2 &= L_1 \frac{n_2^2}{n_1^2} \\ &= 112.5 \times 20^2 = 45000 \mu\text{H} = 45\text{mH} \end{aligned}$$

Flyback Converters



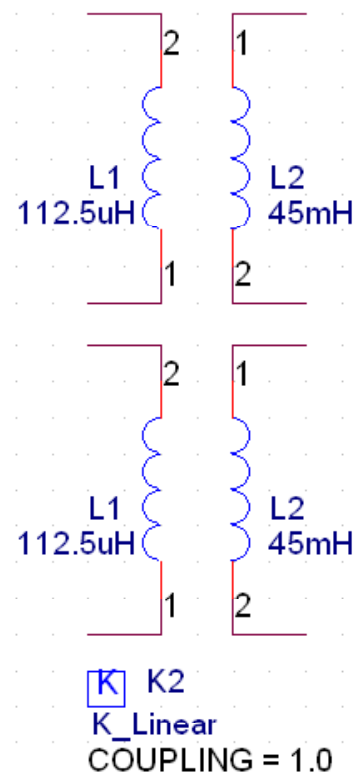
Flyback Converters

Note that the one of the inductors must be inverted:

So far the inductors are separate with no mutual inductance

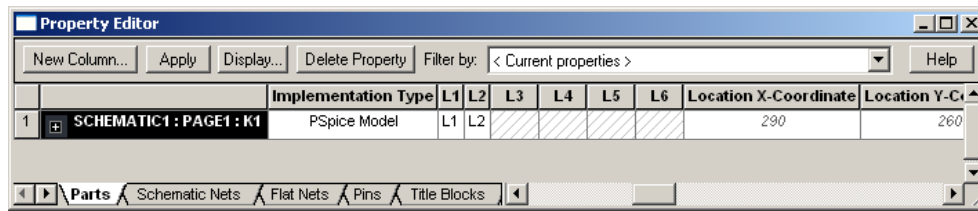
In the Place Part window select: K_Linear from the ANALOG library

Set coupling to 1.0



Flyback Converters

Double click on K to bring up the property editor:



Select the Parts tab, and edit the fields L1 and L2 to contain the identifiers of your two inductors

Set the gate drive parameters to 50% duty cycle at 5 kHz:

V1 = 0.0 V2 = 8.0
TD = 0.0 TR = 10.0n TF = 10.0n
PW = 100.0u PER = 200.0u

Flyback Converters

Place voltage markers on the transformer primary and secondary, and on the output

Place current markers on the primary and secondary windings of the transformer

Click on: PSpice / Edit Simulation Profile and set:

Run to time = 20.4 ms
Start saving data after = 20.0 ms
Maximum step size = (leave blank)

Measure the output voltage and compare this with the theoretical value

Measure the efficiency of the converter (averaged over 200µs)

Flyback Converters

Measure the peak current in primary winding and compare with the theoretical value

Measure the peak voltage across the MOSFET switch and the peak reverse voltage across the rectifier diode

(Use the voltage difference markers: )

Search the web to find the specifications for the MOSFET switch and rectifier diode

Determine whether the voltages and currents in the MOSFET switch and rectifier diode are within their ratings

Flyback Converters

A flyback converter generates constant power into the load (in this case 8W nominal)

Change the load resistance to 15k Ω and measure the output voltage

Explain this decrease in output voltage

What would be the likely effect of removing the load resistance R_L from this flyback converter?

Flyback Converters

A common use of flyback converters is to charge the energy storage capacitor in electronic flash

The energy stored in a 1400 μ F capacitor charged to 300V (Nikon SB800) is:

$$E = \frac{1}{2}CV^2 = \frac{1}{2} \times 1400 \times 10^{-6} \times 300^2 = 63\text{J}$$

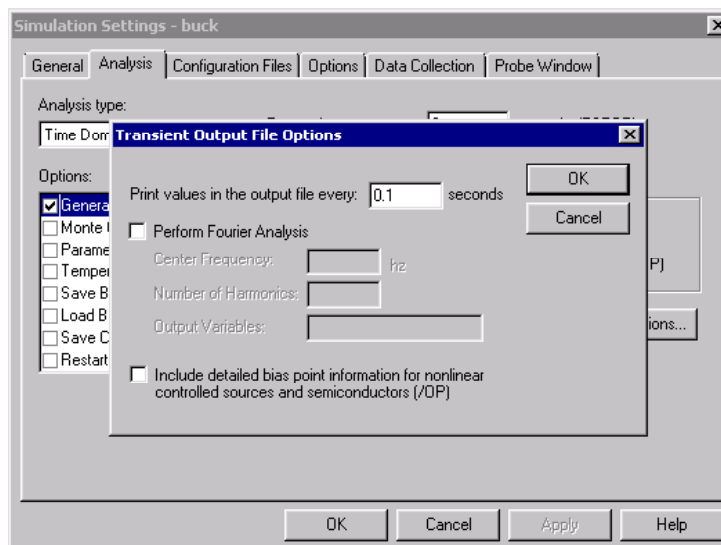
Change the output capacitor on your schematic to 1400 μ F and remove the load resistance

Simulate the flyback converter over the time 0-8s and determine the time for the output voltage to reach 300V

Simulating a 5 kHz converter over 8 s generates a huge amount of data which may exceed storage capacity

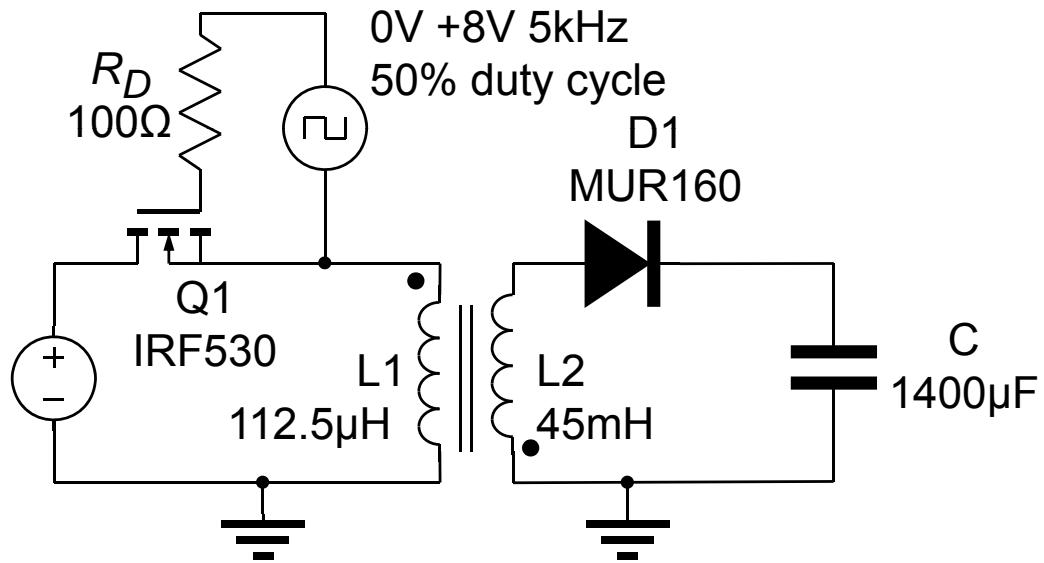
Flyback Converters

To avoid problems when simulating, edit the simulation profile and click: Output file options: set Print values every 0.1 s



Measure the time taken for the capacitor voltage to reach 300 V and compare with calculated value

Flyback Converters



Analogue Simulation

Session 8

DC-DC Converters
Forward Converters

Forward Converters

In flyback converters energy is stored in the transformer's inductance each cycle, and then transferred to the output

Power is therefore limited by the energy storage and the operating frequency

Consequently flyback converters are limited to low-power applications

Forward converters do not rely on energy storage and are suitable for high-power applications

A typical use of forward converters is the 300W power supply in a personal computer

Forward Converters

Forward converters operate in discontinuous mode - that is the primary current falls to zero each cycle

This is achieved by keeping the duty cycle below 50%:
typically $k=0.4$

An arrangement of MOSFET switches and diodes is used to generate a $\pm v_s$ rectangular wave at the transformer primary

The secondary rectangular wave is rectified and filtered by an inductor/capacitor filter

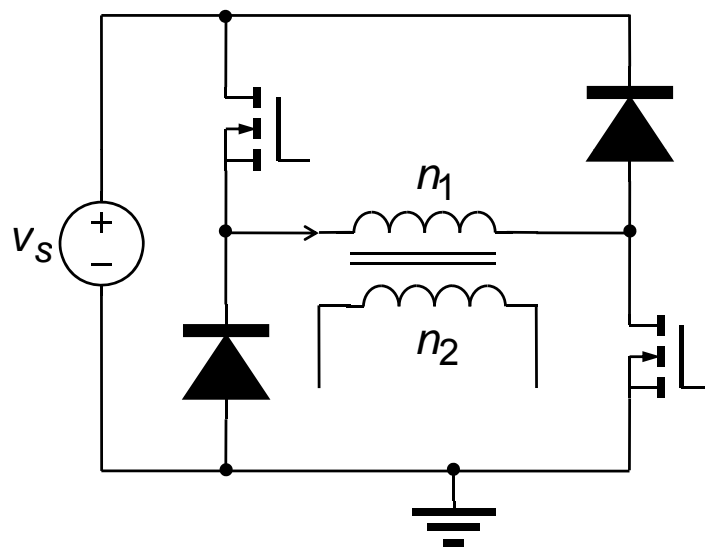
The output voltage is determined by the transformer turns ratio and the duty cycle

Forward Converters

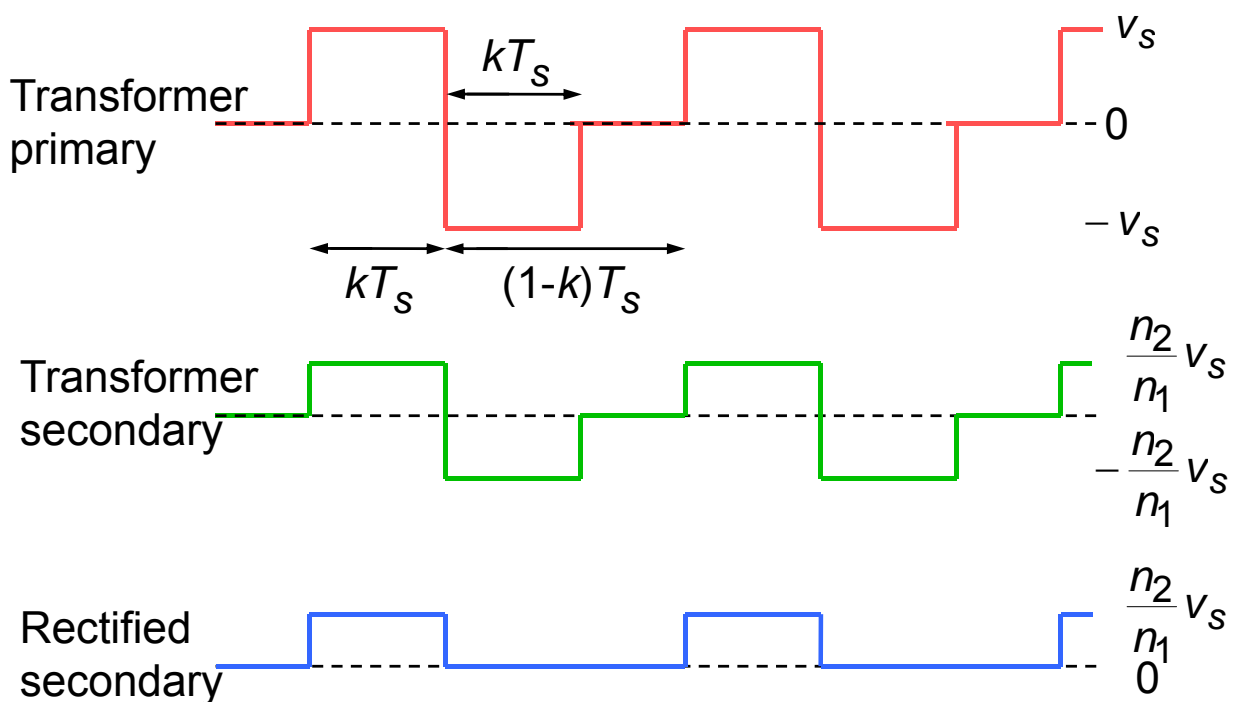
Transistors turn on and off together

When transistors turn off current continues to flow through diodes

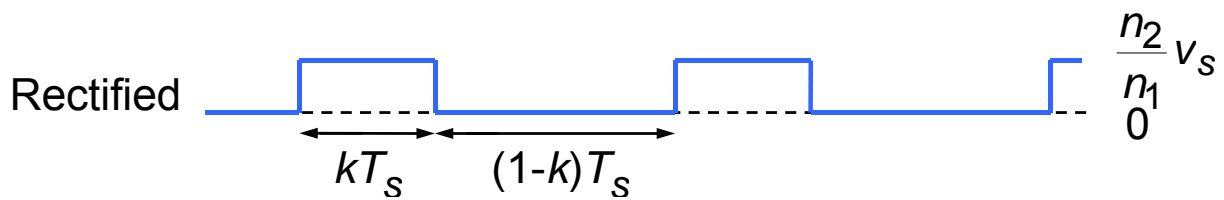
This reverses voltage across primary winding



Forward Converters



Forward Converters



Average output voltage:

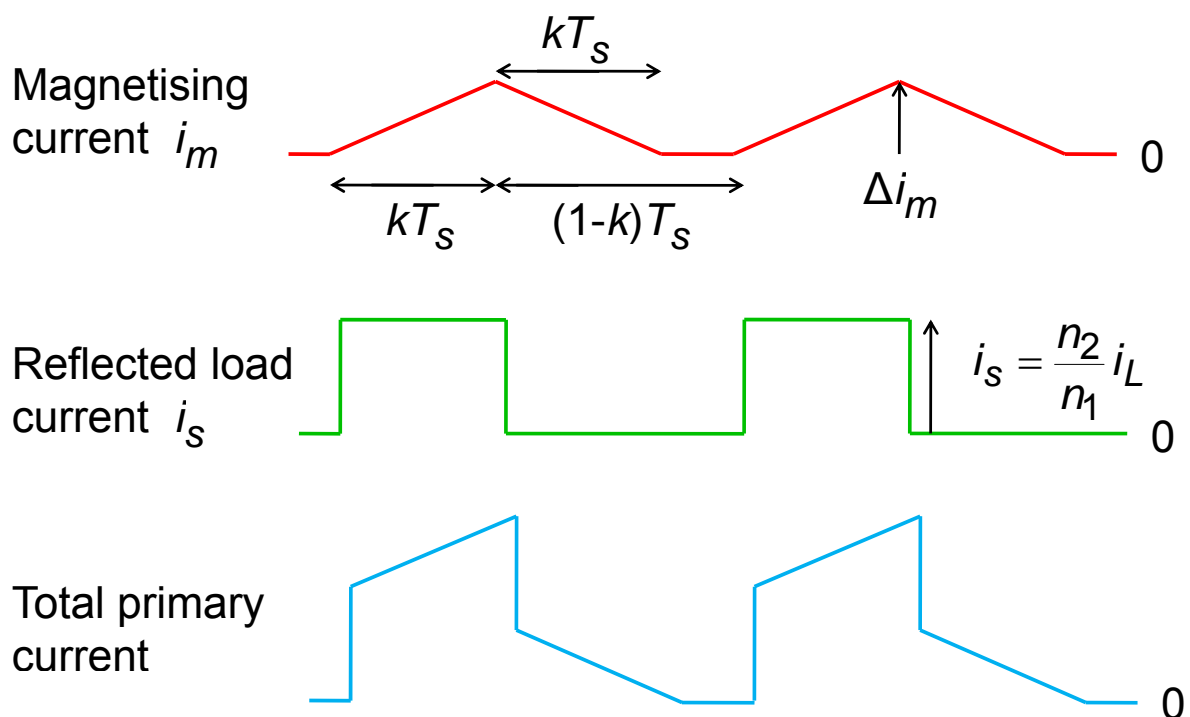
$$v_o = \frac{1}{T_s} \left\{ kT_s \times \frac{n_2}{n_1} v_s + (1-k)T_s \times 0 \right\}$$

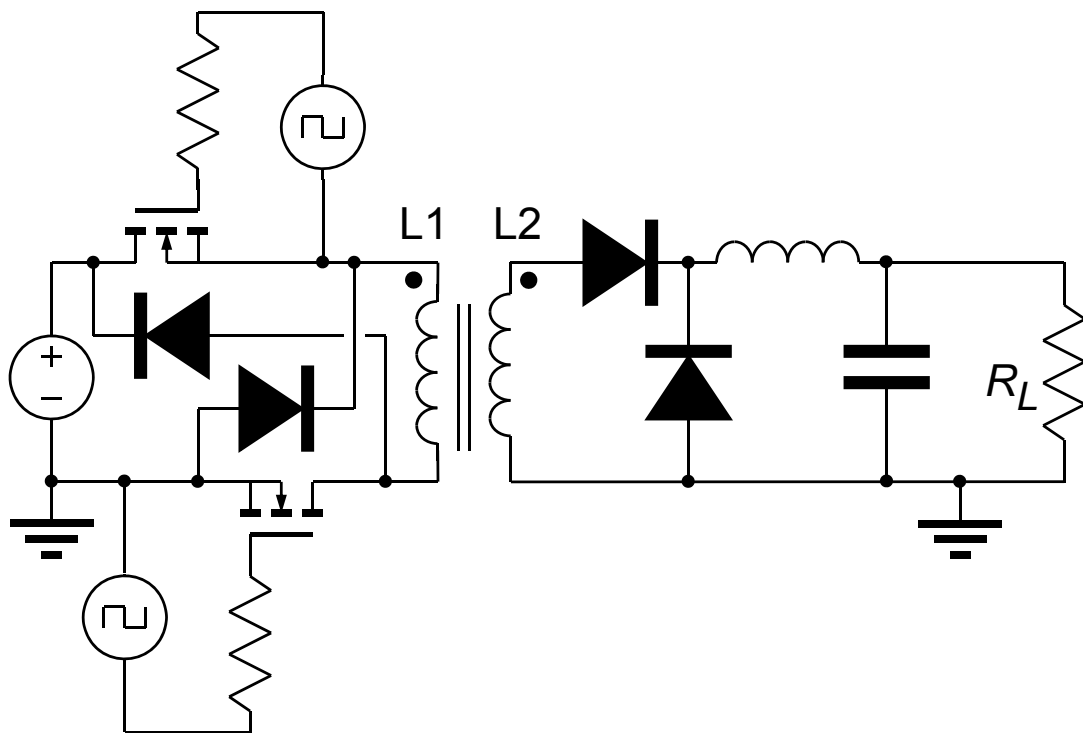
$$= k \frac{n_2}{n_1} v_s$$

Note that secondary transformer winding is not inverted

Forward Converters

Transformer primary current:





Forward Converters

Example: a 50kHz forward converter with a 330V input generating a 5V output into a 0.125Ω load (40A, 200W)

Average input current:

$$\bar{i}_s = \frac{200}{330} = 0.6A$$

Peak current for $k = 0.4$:

$$i_s = \frac{\bar{i}_s}{k} = \frac{0.6}{0.4} = 1.5A$$

Let magnetising current: $i_m = i_s/10 = 0.15A$, so:

$$L_1 = \frac{v_s \times k T_s}{i_m} = \frac{330 \times 0.4 \times 20 \times 10^{-6}}{0.15} = 17.6mH$$

Forward Converters

Transformer turns ratio (assuming 0.6V drop in output rectifier diodes):

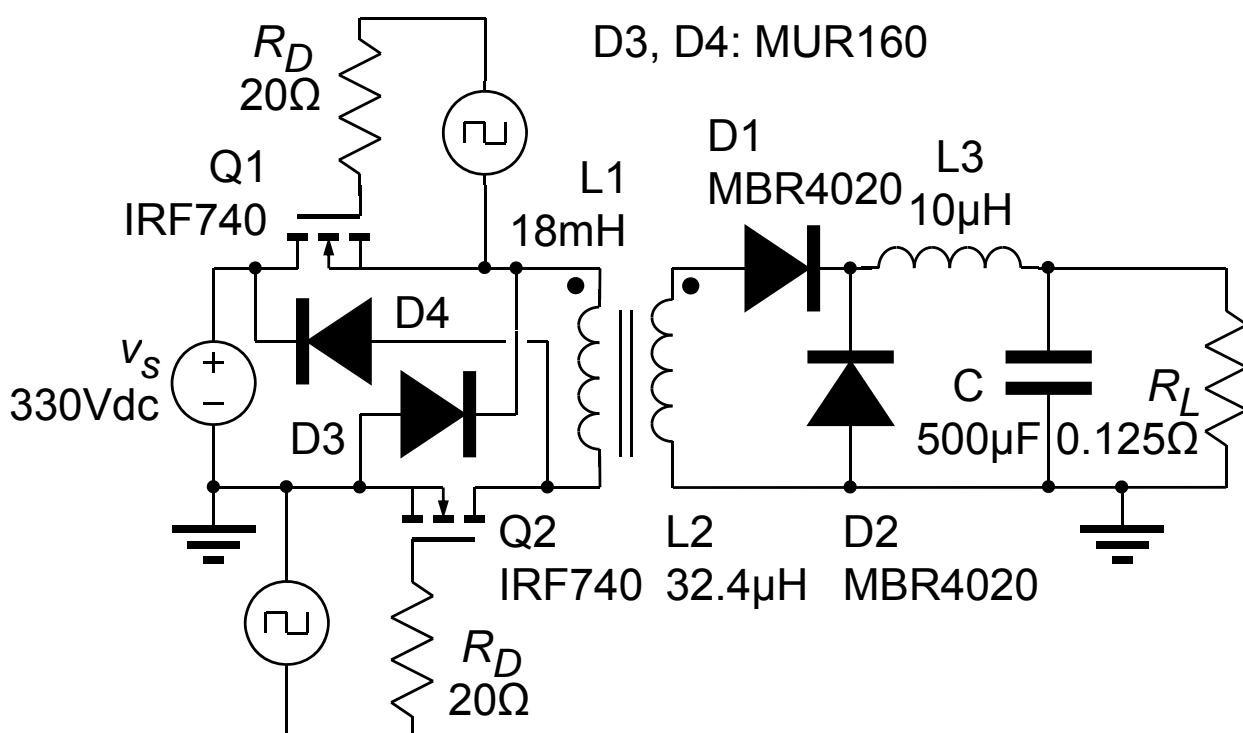
$$v_o = k \frac{n_2}{n_1} v_s$$

$$\frac{n_2}{n_1} = \frac{v_o}{k v_s} = \frac{5.6}{0.4 \times 330} = 0.0424$$

Secondary inductance:

$$L_2 = L_1 \left\{ \frac{n_2}{n_1} \right\}^2 = 18 \times 0.0424^2 = 0.0324 \text{mH}$$

Forward Converters



Forward Converters

The MOSFETs in the forward converter are turned on and off together

For the base drivers of the MOSFETS select from SOURCE: VPULSE and set the parameters:

$$V1 = 0.0$$

$$V2 = 8.0$$

$$TD = 0.0$$

$$TR = 10.0\text{n}$$


$$TF = 10.0\text{n}$$

$$PW = 8.0\text{u}$$

$$PER = 20.0\text{u}$$

Forward Converters

Put voltage markers on the transformer secondary, on the junction of the diodes and on the output

Put voltage difference markers:  across the transformer primary

Put current markers on the transformer primary and secondary

Click on PSpice / Edit Simulation Profile and set:

$$\text{Run to time} = 2040 \mu\text{s}$$

$$\text{Start saving data after} = 2000 \mu\text{s}$$

$$\text{Maximum step size} = 0.1 \mu\text{s}$$

Simulate the forward converter

Forward Converters

Inspect the voltage and current waveforms

Determine the output voltage and output ripple

Measure the efficiency of the forward converter

Measure the voltages across the MOSFETs and diodes and check that these are within the device specifications

Measure the currents in the MOSFETS and diodes and check that these are within the device specifications

Why is the efficiency of this converter lower than that of the other types of converter that have been considered?

Forward Converters

The output voltage of a forward converter is controlled by the duty cycle

The initial simulations were performed using a duty cycle $k=0.4$ which should have produced an output voltage of 5V

The output voltage of a forward converter depends on the duty cycle

Change the duty cycle k to 0.3 and measure the new output voltage

How does this compare with the theoretical value?

Forward Converters

There are advantages in using higher switching frequencies

Higher frequencies allow lower value inductors and capacitors to be used

The disadvantage is that switching losses in the MOSFETs and diodes are proportional to frequency

This leads to a lower overall efficiency at higher switching frequencies

Measure the efficiency at a switching frequency of 200kHz (reduce all capacitor and inductor values by a factor of 4)

Analogue Simulation

Session 9

DC-DC Converters
Current-Control Buck Converters
Hysteretic Buck Converters

Current-Control Buck Converters

Buck converters that are required to generate a constant output voltage have feedback to control the duty cycle

Some applications require a constant output current rather than a constant output voltage

Examples are drivers for stepping motors, electromagnets, arc welding and LEDs

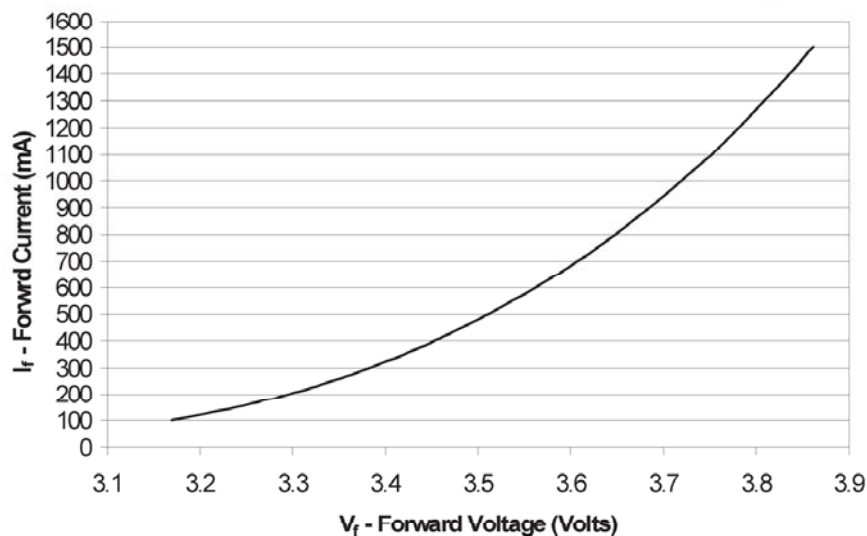
Hysteretic control is a very simple method for controlling the load current of a buck converter

Hysteretic buck converters operate in continuous mode

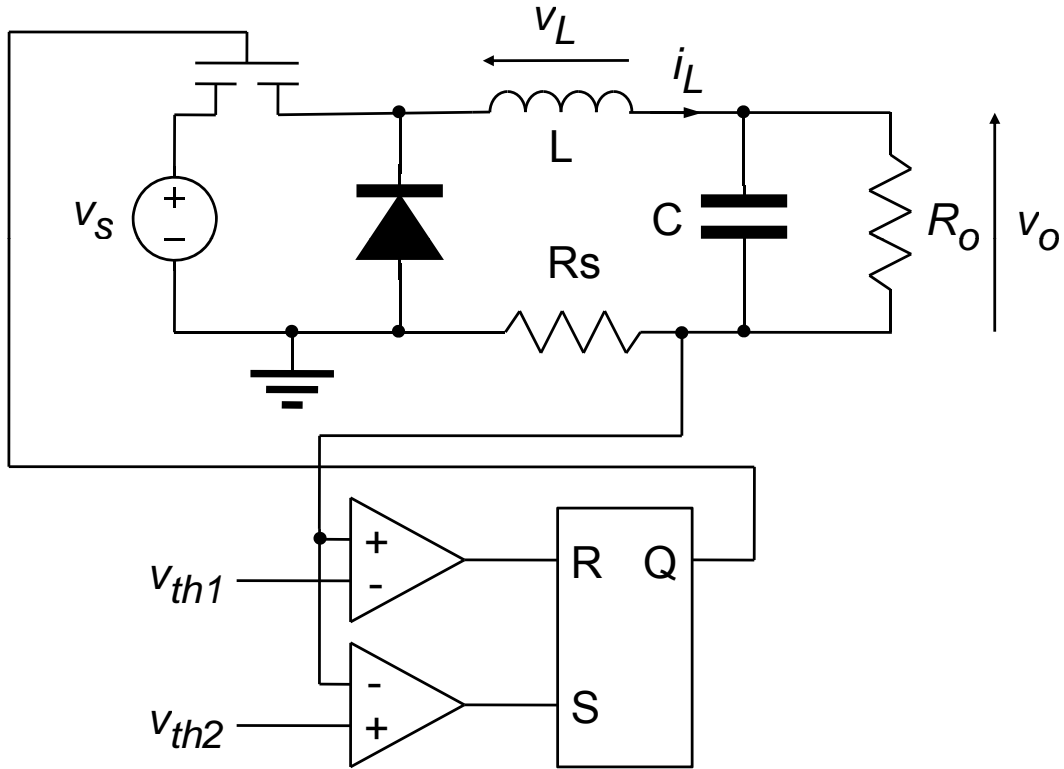
Luxeon K2 Power LED

Typical Forward Current Characteristics

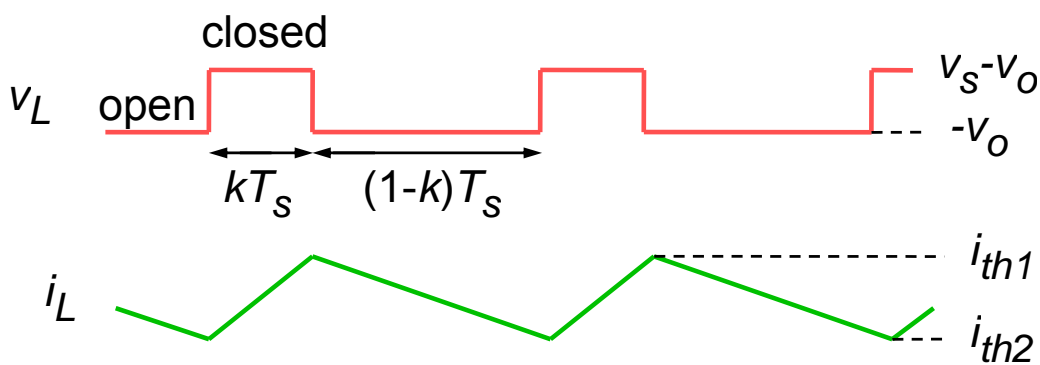
Cool-White, Neutral-White, Warm-White, Green, Cyan, Blue and Royal Blue, Junction Temperature, $T_J = 25^\circ\text{C}$



Current-Control Buck Converters



Current-Control Buck Converters



$$L \frac{di_L}{dt} = -v_o$$

$$L \frac{di_L}{dt} = v_s - v_o$$

$$i_{th2} - i_{th1} = (1 - k)T_s \frac{-v_o}{L}$$

$$i_{th1} - i_{th2} = kT_s \frac{v_s - v_o}{L}$$

Current-Control Buck Converters

$$i_{th2} - i_{th1} = (1 - k)T_s \frac{-V_o}{L}$$

$$i_{th1} - i_{th2} = kT_s \frac{V_s - V_o}{L}$$

$$(i_{th1} - i_{th2}) \frac{L}{V_o} = (1 - k)T_s$$

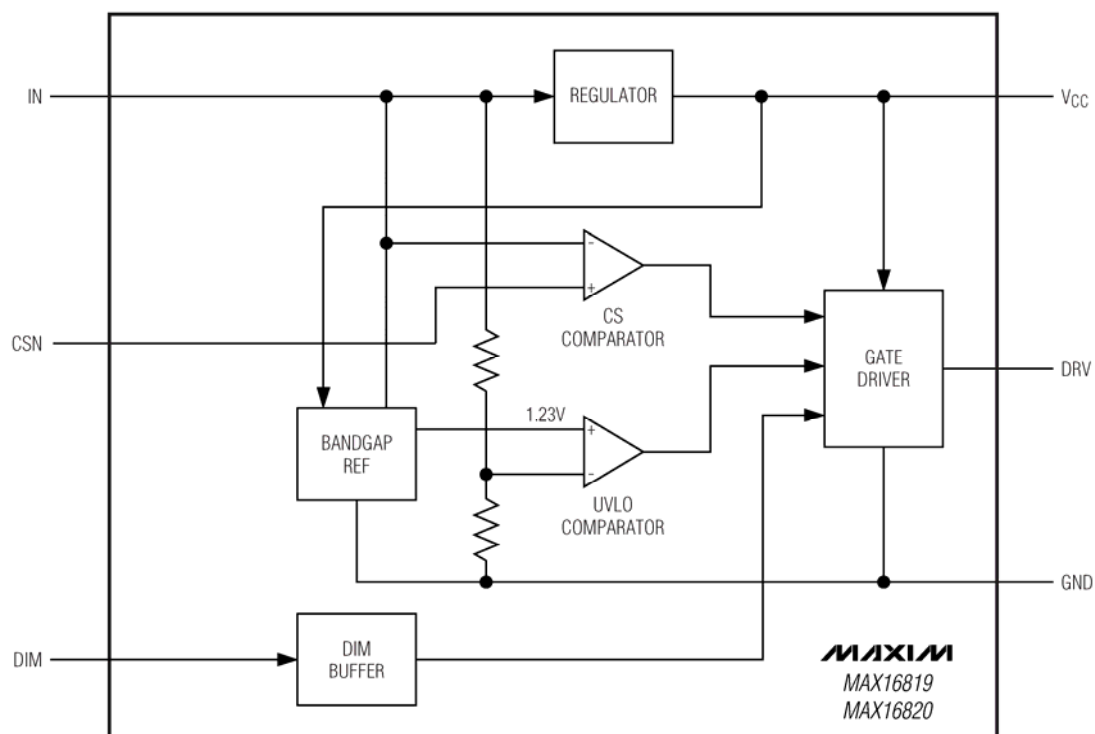
$$(i_{th1} - i_{th2}) \frac{L}{V_s - V_o} = kT_s$$

$$(i_{th1} - i_{th2}) \frac{L}{V_s - V_o} + (i_{th1} - i_{th2}) \frac{L}{V_o} = T_s$$

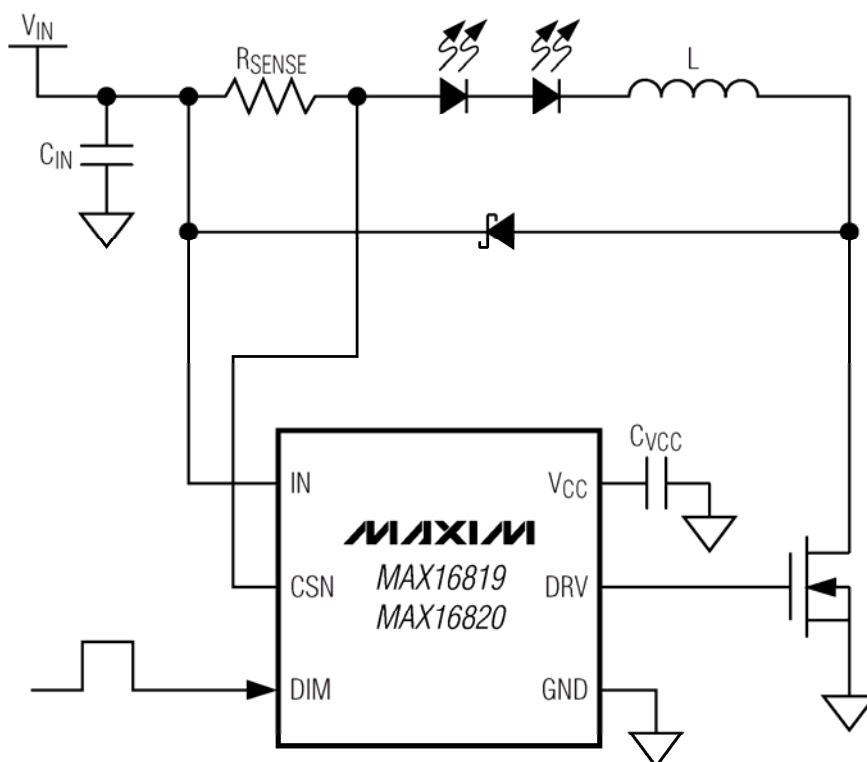
$$T_s = L(i_{th1} - i_{th2}) \frac{V_o + V_s - V_o}{V_o(V_s - V_o)}$$

$$T_s = \frac{L(i_{th1} - i_{th2})}{V_o(1 - V_o/V_s)}$$

Maxim MAX16820



Maxim MAX16820



Maxim MAX16820

ELECTRICAL CHARACTERISTICS

PARAMETER	SYMBOL	MIN	TYP	MAX	UNITS
Input Voltage Range	V _{IN}	4.5		28.0	V
Maximum Current Regulator Switching Frequency	f _{sw}			2	MHz
Ground Current	I _{GND}			1.5	mA
Supply Current	I _{IN}			425	μA
Undervoltage Lockout	UVLO		4.7	5.0	V
Undervoltage Lockout Hysteresis			0.5		V

SENSE COMPARATOR

PARAMETER	SYMBOL	MIN	TYP	MAX	UNITS
Sense Voltage Threshold High	V _{NSHI}	195	210	225	mV
Sense Voltage Threshold Low	V _{NSLO}	176	190	204	mV
Propagation Delay to Output High	t _{DPDH}		82		ns
Propagation Delay to Output Low	t _{DPDL}		82		ns
Current-Sense Input Current	I _{CSN}			1	μA
Current-Sense Threshold	C _{SHYS}		17	35	mV

Design of LED Controller

Supply voltage = 12V

Choose LED current = 800mA

From data sheet LED voltage = 3.73V

MAX16820 threshold voltages = 190mV and 210mV

MAX16820 mean threshold = 200mV

Use sense resistor $R_{sense} = 200\text{mV} / 800\text{mA} = 0.25\Omega$

Threshold currents = 760mA and 840mA

Threshold differential = 80mA

Choose $f_s = 100\text{kHz} \rightarrow T_s = 10\mu\text{s}$

$$L = T_s \frac{V_o(1 - V_o/V_s)}{(i_{th1} - i_{th2})} = 10 \times 10^{-6} \times \frac{7.46(1 - 7.46/12)}{0.08} = 353\mu\text{H}$$

Create New Orcad Library

To simulate a LED driver circuit based on the MAX16820 two components are required that are not in the Orcad Library:

MAX16820 led driver
Luxeon K2 LED

Create a new directory: Lib and extract the files LuxeonK2.lib and MAX16820.LIB from spice_library.zip

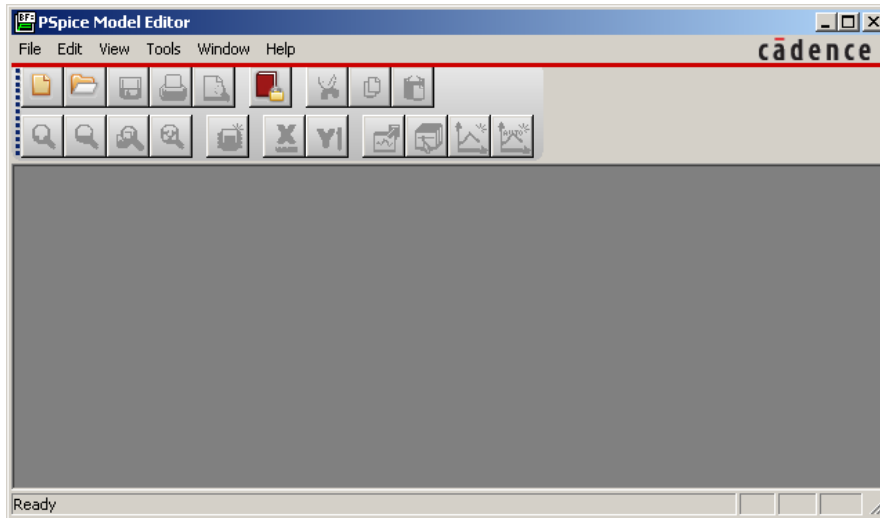
These files are standard SPICE models - open LuxeonK2.lib with Notepad and examine its contents

Without closing LuxeonK2.lib, open MAX16820.LIB in Notepad, and cut/paste into LuxeonK2.lib

Save the combined file which now contains the two SPICE models as MyLib.lib

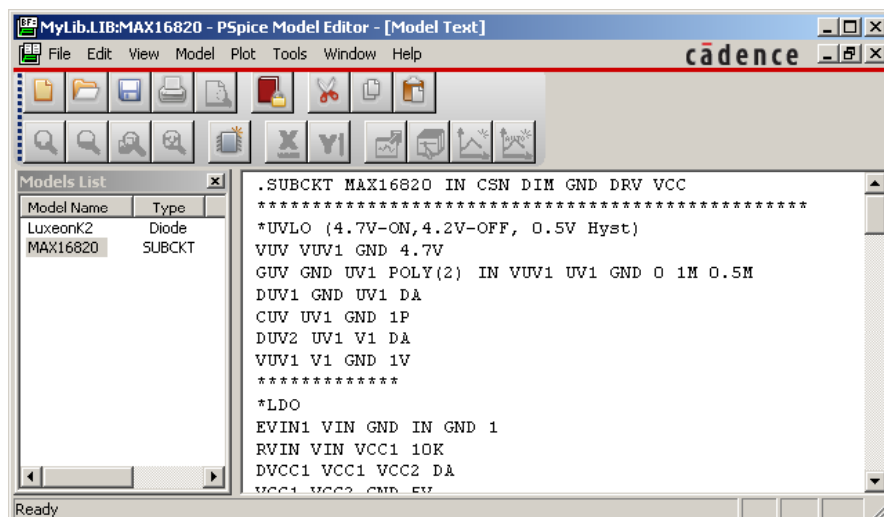
Create New Orcad Library

On desktop click on: Start / Programs / Orcad16.0 / Pspice Accessories / Model Editor



Click: File / Open / MyLib.lib

Create New Orcad Library



Click on: File / Export to Part Capture Library

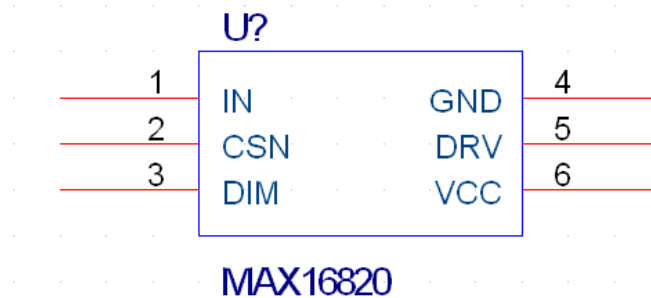
Save as: MyLib.olb and close the PSpice model editor

Create New Orcad Library

In Orcad Capture select: File / Open / Library and browse to MyLib.olb

In the window expand: mylib.olb and you will see the new Orcad parts that you have created: LUXEONK2, MAX16820

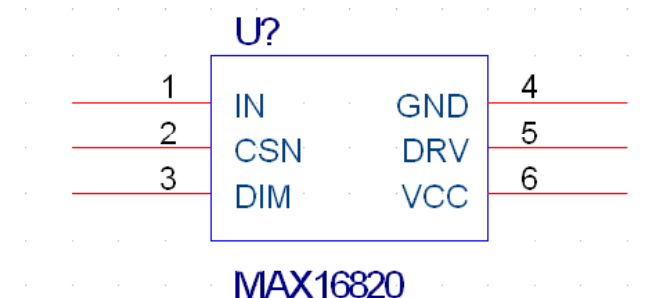
Click on: MAX16820



If require this part can be edited

Create New Orcad Library

Click on the bottom right-hand corner of the outline to resize:

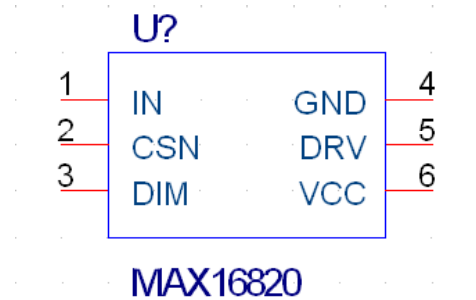


To move the one of the pins, click on it and drag it to the required location (not necessary in this case)

To change the appearance of a pin click on it, then right click and select: Edit Properties / Shape / Short

Create New Orcad Library

MAX16820 circuit symbol with short pins:



Do not change the pin names

Finally click: File / Save, File / Close, File / Close Project and select: Yes All

You are now ready to use this part in a PSpice simulation

Simulate LED Driver

Enter the circuit diagram for the MAX16820 hysteretic converter into ORCAD Capture using the new device models

For the MOSFET use device IRF530

For the flywheel diode use IN5817 (D1N5817)

For Cvcc use a 47nF capacitor

Connect the DIM input of the MAX16820 to the positive supply

Simulate the circuit from 0 to 100 μ s

You will now see the error messages:

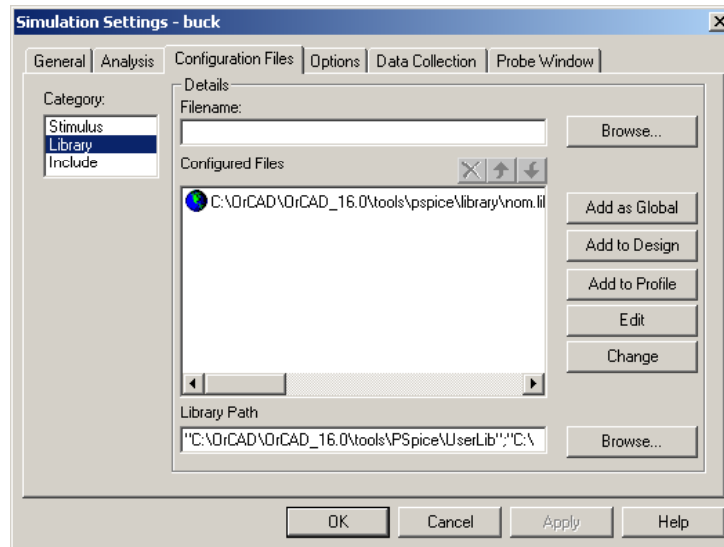
ERROR -- Model LUXEONK2 used by D4 is undefined
ERROR -- Model LUXEONK2 used by D5 is undefined
ERROR -- Subcircuit MAX16820 used by U1 is undefined

Orcad needs to be given the locations of the device models

Simulate LED Driver

Click on: PSpice / Edit Simulation Profile and select the Configuration Files tab

Under: Category select: Library



Simulate LED Driver

Browse to the device models: MyLib.lib and click on: Add as Global

Now select the Analysis tab and tick the box: Skip the initial transient bias point calculation

Click on: OK

Place a voltage marker on the drain of the MOSFET

Place current markers on one of the LEDs and on the flywheel diode

Simulate the circuit from 0 to 100 μ s

Simulate LED Driver

Click on: PSpice / Edit Simulation Profile and set:

Run to time = 100 μ s

Start saving data after = 80 μ s

Maximum step size = 0.1 μ s

Inspect the current waveforms

Measure the minimum and maximum LED currents and calculate the average current

How do these currents compare with the design values?

Efficiency of LED Driver

In order to evaluate the efficiency we need to know the exact period T_s of the waveform

In the simulation window click on: Trace / Measurements, select: Period from the list and click on: Eval

In Name of trace to search select the voltage on the MOSFET drain: V(M1:d) and click on: OK

The period is now displayed near the top of the window

Use this value of T_s to average the power from the supply and the power in the LEDs, and hence calculate the efficiency

Compare this value of T_s with the design value

Simulate LED Driver

The battery voltage in a car varies from 12V (partially charged) to 14V (fully charged)

Ideally the current in the LEDs should be independent of the supply voltage

Change the supply voltage to 14V and simulate the converter

Measure the minimum and maximum LED currents and calculate the average current

Are these currents significantly different from those measured with a supply voltage of 12V?

Simulate LED Driver

The data sheet for the MAX16820 gives a maximum switching frequency of 2MHz

The advantage of a higher switching speed is a smaller and cheaper inductor but the efficiency is likely to be lower

Calculate the value of inductor that would give a switching speed of 1MHz

Simulate the circuit at this higher switching speed

Measure the actual switching frequency, the output current and the efficiency

Analogue Simulation

© J. B. Grimbleby 21 November 2012