Lab Manual

Power Electronics (EE460)

June - July 2006

Prepared by:

Dr. Mahmoud Kassas

Summary

The EE460 LAB final report documents are the achievement during the lab development. Lab material has been prepared together using Microsoft Word, PSpice, and Visio. The entire lab material has been revised and new lab experiments are added. All experiments have been implemented and tested using the COM3LAB by Leybold.

Table of Contents

COM3LAB BOARD PASSIVE ELEMENTS

The values of the R, L and C's loads used on the Power Electronics Board are:
The data are provided by the HEIP "Al-Harbi for Education & Informational Projects".

C4 = 200 μF
C1-C3 = 2.2 μF
R = 17Ω-170Ω (14V / 80 mA)
L11-L32 = 250 mH (1 kHz test)
L41 = 6 mH (measured 1 kHz test)
L42 = 25 mH (measured 1 kHz test))

The values of the R, L and C's loads used on the Power Electronics Board are:
The data are measured by KFUPM Instructor.

L11:	**R = 811.2Ω & L = 273.7 mH**	**L12:**	**R = 3.237 kΩ & L = 1.195 H**
L21:	**R = 797.1Ω & L = 264.7 mH**	**L22:**	**R = 3.123 kΩ & L = 1.155 H**
L31:	**R = 812.7Ω & L = 265.4 mH**	**L32:**	**R = 3.236 kΩ & L = 1.163 H**
L41:	**R = -2.33Ω & L = 5.317 mH**	**L42:**	**R = 2.807 Ω & L = 21.66 mH**

H11-H12: R = 12.22 Ω
H21-H22: R = 13.73 Ω
H31-H32: R = 13.11 Ω

C1 = 2.165 μF
C2 = 2.146 μF
C3 = 2.241 μF
C4 = 196.2 μF

LAB 1: SINGLE-PHASE HALF-WAVE RECTIFIER - R & RL LOADS

LARNING OBJECTIVES

After mastering this unit you will:

- Know the operation of a single-phase diode rectifier with resistive and inductive loads and their performance parameters.
- Know the effects of a free-wheeling diode on the performance of the rectifier.
- Be able to model a diode rectifier circuit.

REFERENCE: Rashid, *Power Electronics*, Circuits, Devices, & Applications, Third Edition, Pearson – Prentice Hall, 2003.

READING ASIGNMENT

Read section 3.3 in chapter 3 of the text book.

THEORY

The circuit diagram of a single-phase half-wave diode rectifier is shown in Fig. 1-1. The performance parameters of the rectifier with a purely resistive load are given by:

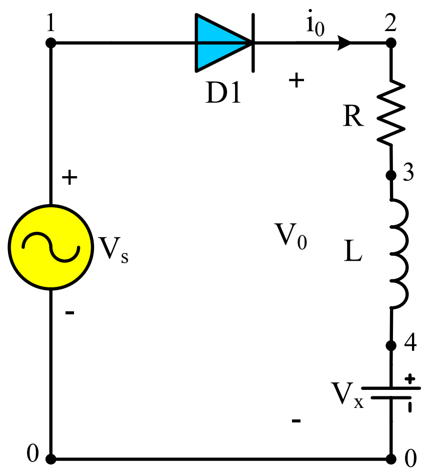


Figure 1-1 Single-phase half-wave rectifier

1. RMS input voltage, V_s.
2. Peak input voltage, $V_m = \sqrt{2}V_s$
3. Supply frequency, f_s.
4. RMS input current, I_s.
5. Average value of the output (load) voltage, $V_{o(dc)} = V_m/\pi = 0.318\ V_m$.
6. Average value of the output (load) current, $I_{o(dc)} = V_{o(dc)}/R = 0.318\ V_m/R$.
7. Output DC power, $P_{o(dc)} = V_{o(dc)}\ I_{o(dc)}$.
8. RMS value of the output voltage, $V_{o(rms)} = V_m/2 = 0.5\ V_m$.
9. RMS value of the output current, $I_{o(rms)} = V_{o(rms)}/R = 0.5\ V_m/R$.
10. Output AC power, $P_{o(ac)} = V_{o(rms)}\ I_{o(rms)}$.

11. RMS value of the ac component of the output voltage, $V_{o(ac)} = \left[V_{o(rms)}^2 - V_{o(rms)}^2\right]^{1/2}$.
12. Ripple factor of the output voltage, $RF_v = V_{o(ac)} / V_{o(dc)}$.
13. RMS value of the ac component of the output current, $I_{o(ac)} = \left[I_{o(rms)}^2 - I_{o(rms)}^2\right]^{1/2}$.
14. Ripple factor of the output voltage, $RF_i = I_{o(ac)} / I_{o(dc)}$.
15. Transformer utilization factor, $TUF = P_{o(dc)}/V_sI_s$.

Input power factor is defined as:

$$PF = \frac{V_s I_{s1}}{V_s I_s}\cos\phi_{s1} = \frac{I_{s1}}{I_s}\cos\phi_{s1}$$

Where: I_{s1} the fundamental component of the input current

ϕ_{s1} the angle between the fundamental components of the input current and the input voltage.

The instantaneous output voltage is given by:

$$v_o(t) = \frac{V_m}{\pi} + \frac{V_m}{2}\sin\omega t - \frac{2V_m}{3\pi}\cos 2\omega t + \frac{2V_m}{15\pi}\cos 4\omega t - \frac{2V_m}{35\pi}\cos 6\omega t + \ldots$$

Where $\omega = 2\pi f_s = 2\pi \times 60 = 377$ rad/sec.

PSpice SIMULATION

Schematics:

1. If you are in the Program Manager of the Window, double click the left mouse button on the Design Center icon to open the Design Center Group (showing Schematics, PSpice, Probe, Parts).
2. Double click the left mouse on the Schematics icon to open the Schematics menu (showing File, Edit, Draw, Analysis, etc.).
3. Open the Draw menu by clicking once on the Draw menu. Choose Get New Parts, and then Browse.
4. Get part an AC sinusoidal source VSIN and a DC battery VDC from the source.slb library. **(Set VSIN amplitude to 11.5 V and VDC to 0V)**
5. Get part resistance R from the analog.slb library.
6. Get part diode D1N4148 from the eval.slb library (you may choose a different diode from the PSpice library).
7. Get part earth ground AGND from the port.slb library.
8. Draw and complete the diode circuit shown in Fig. 1-1. Store the schematic in a file, say d:\EE460\lab1.sch. You can click the left mouse on the device or element and choose Attributes from the Edit menu. Alternately, you can change the attributes of any devices or elements by double clicking the left mouse and typing the new values.
9. Analyze the circuit of Fig. 1-1 by choosing Analyze from the Schematics menu.

- Click once on the Analyze menu and then choose the Setup menu.
- Choose the analysis type - Transient and give the transient information: print step, 10 μs, final time, 100 ms, No-Print Delay, 66.66ms, Step Ceiling, 10 us, and Fourier information: center frequency, 60 Hz, No. of harmonics 10, and output variable, I(VX) V(R).
- Run the simulation by choosing Simulate from the Analyze menu.

- After successful simulation, PSpice will automatically run Probe and move to Probe menu. Chose Add from the Trace menu of Probe and choose the plot variable, the output current, e.g., I(R).
  The PSpice plots of the output voltage V(R:1) and the input voltage V(VS:+) are shown in Fig. 1-2. Figure 1-3 shows the Fourier analysis setting. The plots of the rms output current RMS(I(VX)) and the average output current AVG(I(VX)) are shown in Fig. 1-4. Note that the Level and Cursor are activated from the Tools menu of Probe.

- After the completion of the simulation, PSpice stores the results of Fourier analysis on the output file, d:\EE460\lab1.out. You can view the output file from the File menu of PSpice. (See sample of some Fourier analysis data in the output file below)

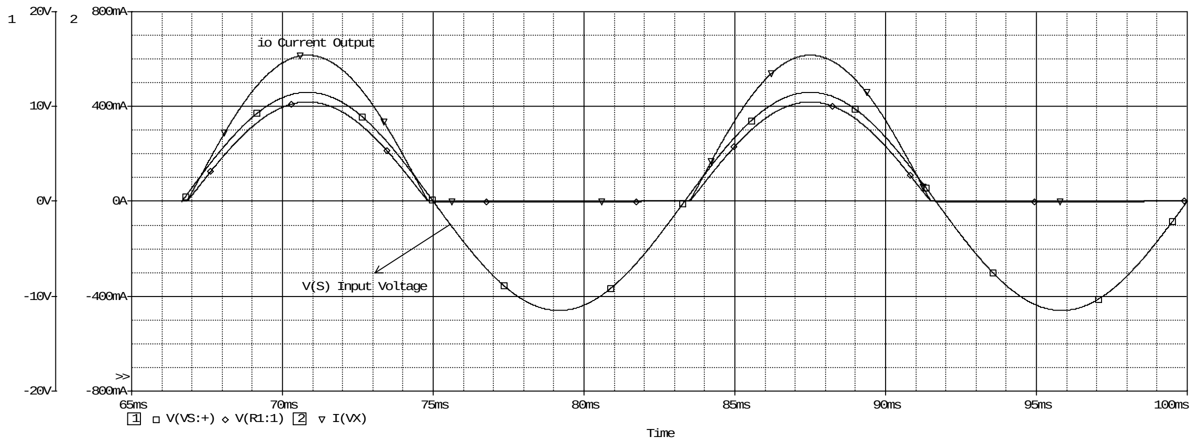


Figure 1-2 Output voltage and current waveforms of half-wave rectifier

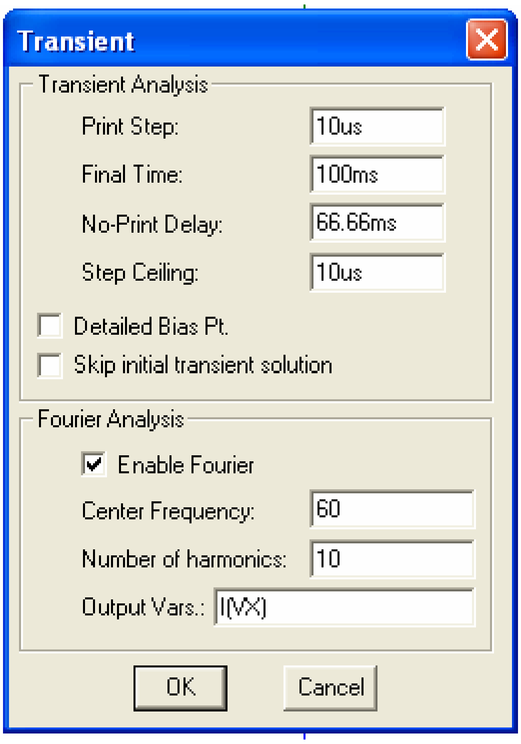


Figure 1-3 Fourier analysis setting

FOURIER COMPONENTS OF TRANSIENT RESPONSE I(V_VX)

DC COMPONENT = 1.878734E-01

HARMONIC NO	FREQUENCY (HZ)	FOURIER COMPONENT	NORMALIZED COMPONENT	PHASE (DEG)	NORMALIZED PHASE (DEG)
1	6.000E+01	3.013E-01	1.000E+00	-6.476E-06	0.000E+00
2	1.200E+02	1.398E-01	4.639E-01	-9.000E+01	-9.000E+01
3	1.800E+02	1.050E-02	3.483E-02	1.800E+02	1.800E+02
4	2.400E+02	2.650E-02	8.793E-02	-9.000E+01	-9.000E+01
5	3.000E+02	5.829E-03	1.934E-02	1.800E+02	1.800E+02
6	3.600E+02	1.060E-02	3.517E-02	-9.000E+01	-9.000E+01
7	4.200E+02	3.897E-03	1.293E-02	1.800E+02	1.800E+02
8	4.800E+02	5.400E-03	1.792E-02	-9.000E+01	-9.000E+01
9	5.400E+02	2.848E-03	9.451E-03	1.800E+02	1.800E+02
10	6.000E+02	3.089E-03	1.025E-02	-9.000E+01	-9.000E+01

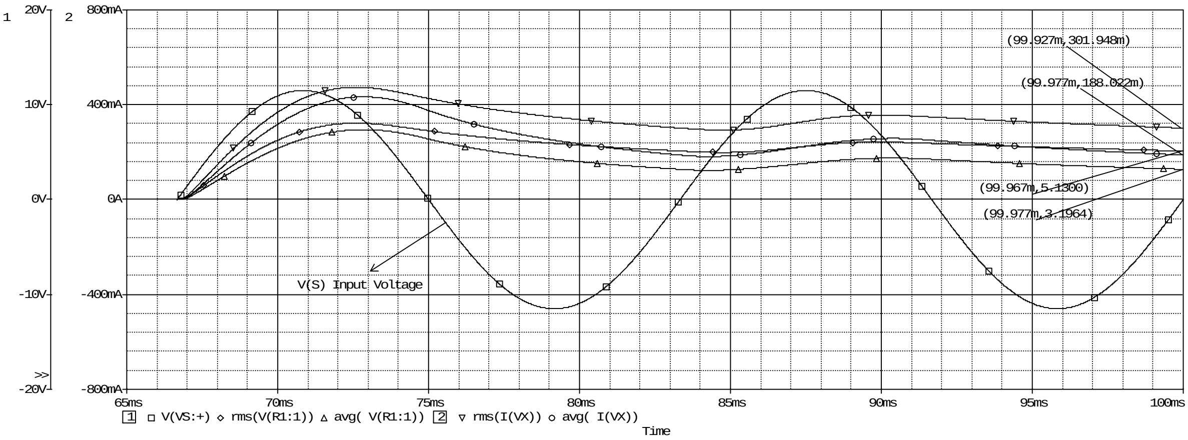


Figure 1-4 Calculation of RMS & AVG output voltage and current

OR, Go to the file menu of Pspice, choose new and give a file name. Type the following statements:

Type your name, ID #, Experiment #, Course # and Date in the first line or leave it blank

```
VS 1   0   SIN (0   11.5   60HZ)
R   2   3   17
L   3   4   0.0
VX4   0   DC 0V ; Voltage source to measure the output current
D1 1   2   DMOD
.MODEL   DMOD  D (IS=2.22E-15 BV=1200V IBV=13E03 Cjo=2PF TT=1US)
.TRAN    10US 50.0MS 16.66MS 10US ; Transient analysis
.PROBE                      ; Graphics post processor
.FOUR    60HZ   V(2) I(VX)
.OPTIONS ABSTOL = 1.0N RELTOL = 0.01 VNTOL = 1.0M ITL5 = 0; Convergence
.END
```

Save the file. **(make sure the extension of the file is filename.cir)**

Run analysis

Plot V(2), I(VX), V(1,2) = (V(1)-V(2))

Figure 1-5 shows the voltage and current output for the RL load for the circuit without freewheeling diode.

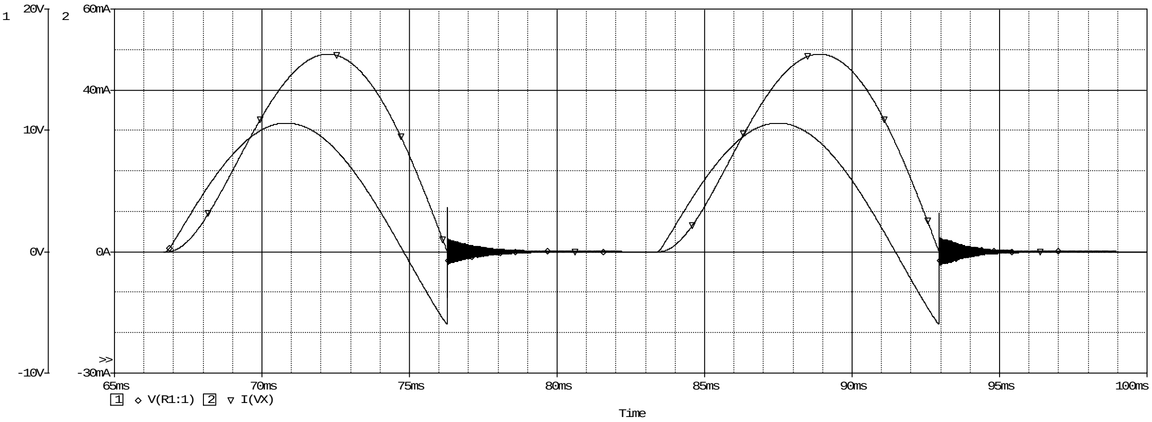


Figure 1-5 Voltage and current output for RL load without freewheeling diode

RESULTS

1. Run the simulation of Fig. 1-1 and complete the Table 1-1 for two values of load inductances.

Table 1-1 Single-phase half-wave rectifier without freewheeling diode

L	R	$V_{o(dc)}$	$I_{o(dc)}$	$V_{o(rms)}$	$I_{o(rms)}$	$THD_{i(input)}$	PF
0	15Ω						
L42=21.3mH	17Ω						
L11=313mH	185Ω						

2. A free-wheeling diode Dm is normally connected across the inductive load to smooth the output current, thereby improving the input power factor. This is shown in Fig. 1-6. The output current I(VX), the free-wheeling current I(Dm) and the input current –I(VS) are shown in Fig. 1-7. Run the simulation of Fig. 1-6 and complete Table 1-2 for two values of load inductances.

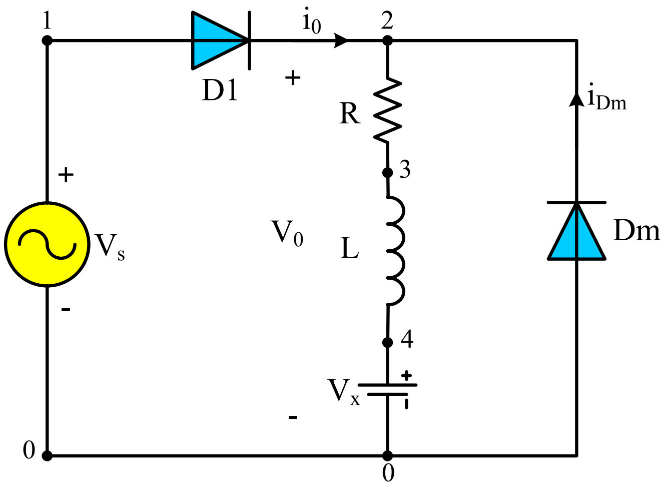


Figure 1-6 Single-phase half-wave rectifier with free-wheeling diode

Table 1-2 Single-phase half-wave rectifier with free-wheeling diode

L	R	$V_{o(dc)}$	$I_{o(dc)}$	$V_{o(rms)}$	$I_{o(rms)}$	$THD_{i(input)}$	PF
L42=21.3mH	17Ω						
L11=313mH	185Ω						

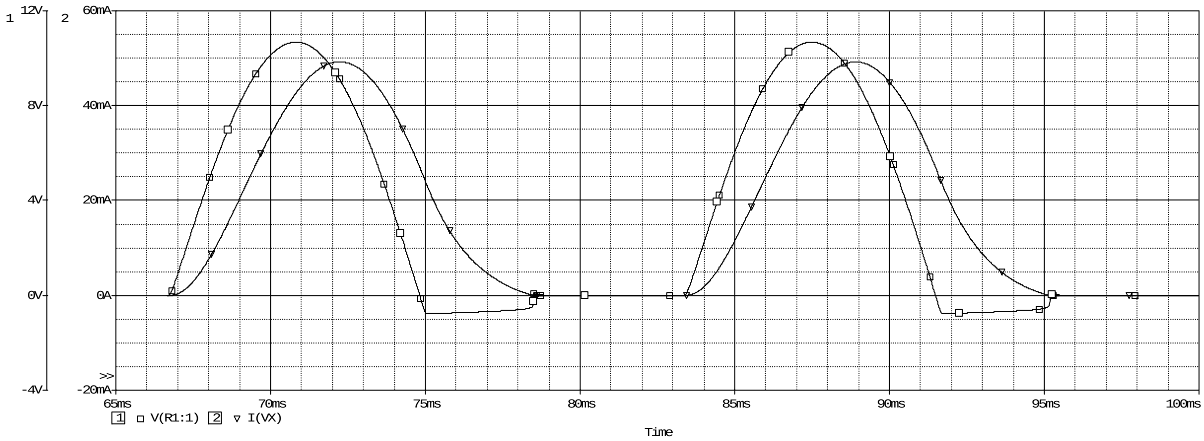


Figure 1-7 voltage and current waveforms of half-wave rectifier

DISCUSSION

You presented with the PSpice modeling and simulation of a single-phase half-wave rectifier. Although, the half-wave rectifier is not normally used in practical power applications, it gives you ideas about the principle of operation of rectifiers in general and the functional of free-wheeling diodes.

Key point of the laboratory:

1. An inductive load causes a delay of the diode current which extends beyond the zero-crossing of the input voltage.
2. A free-wheeling diode provides a path for the continuity of the inductive load current during the negative half-cycle of the input voltage. Thus, it smoothes out the load current and improves the input power factor.
3. The output voltage contains harmonics. An output filter is normally used to obtain a smooth DC output voltage. The load inductance L acts as the output filter also.
4. Also, the input current contains harmonics. An input filter is normally used to block harmonic currents flowing into the supply line.
5. The input power factor is dependent upon the load impedance.
6. With an inductive load, the conduction of the diode extends beyond 180o. The diode stops conducting when its current has fallen to zero, but not when the input voltage reverses its polarity. With a very highly inductive, the load current my not go through zero and the diode current would be continuous. That is, the diode would conduct continuously.

EXPERIMENT

Uncontrolled single-pulse rectifier under resistive load

After completing the connection of the above circuit you can run the experiment as the following:

Turn **ON** the board.

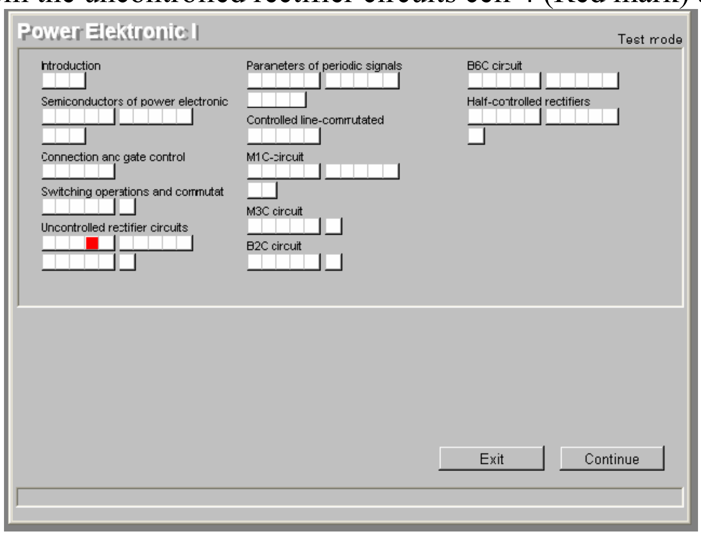
Click → **Start button** → **Programs** → **COM3LAB** → **70021 power electronic I**.

Window will open, type your initial name (such as "**mk**") and then click **Continue**.

Click again Continue.

1. 

Click content (Overview/Exist) to select the experiment.
2. Select from the uncontrolled rectifier circuits cell 4 (Red mark) and click.



3. A new window will open with the circuit as the following:

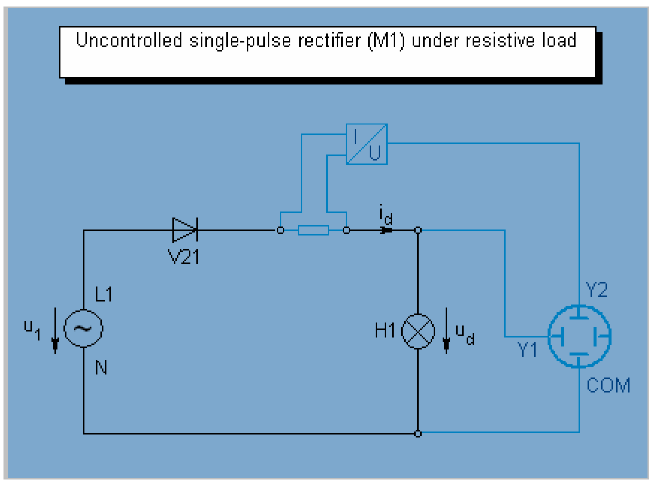


Figure 1-8 Uncontrolled single-pulse rectifier under resistive load

From the board of **COM3LAB 70021** Power Electronics do the following:

1. Connect the source terminal **L1** to diode anode **V21**.

2. Connect the diode cathode **V21** to the shunt resistor top terminal in the bottom of the board.
3. Connect the bottom terminal of the shunt resistor to resistive load terminal **H11** (light bulb).
4. Connect the terminal **H12** (light bulb) to the neutral of the source **N**.
5. Connect the output of the shunt resistance to **Y2** of the oscilloscope (on the board).
6. Connect the terminal **H11** to **Y1** of the oscilloscope (on the board).
7. Connect the terminal **H12** to **COM** of the oscilloscope (on the board).
8. Click first on the **oscilloscope** button then on the **oscilloscope panel** button.
9. From the oscilloscope panel, change the setting of **Y2/div** from **5V** to **1V** and the Curve Y1 to Curve **DUAL**.
10. Click **Run** on the panel to display the voltage and current waveforms as seen in Figure 1-9.

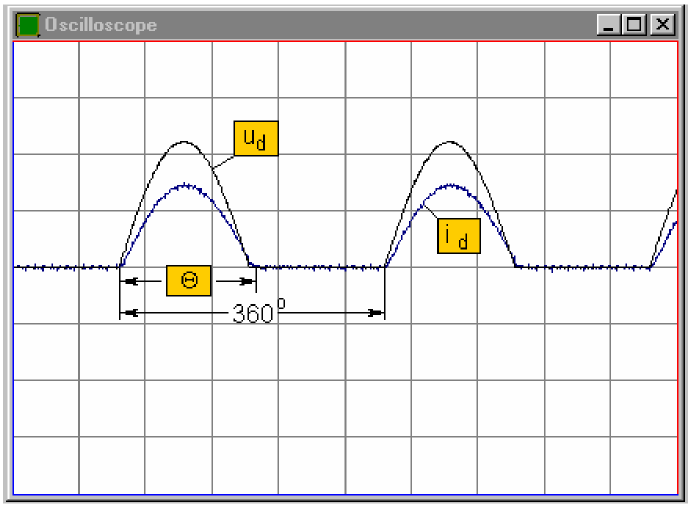


Figure 1-9 Single-phase half-wave rectifier for resistive load

11. Repeat the experiment for the R & L values are shown in Table 1-1. Fig. 1-10 shows the voltage and current output waveforms for RL-Load without freewheeling diode
12. Repeat the experiment for the R & L values are shown in Table 1-2 with freewheeling diode as shown in Fig. 1-11.

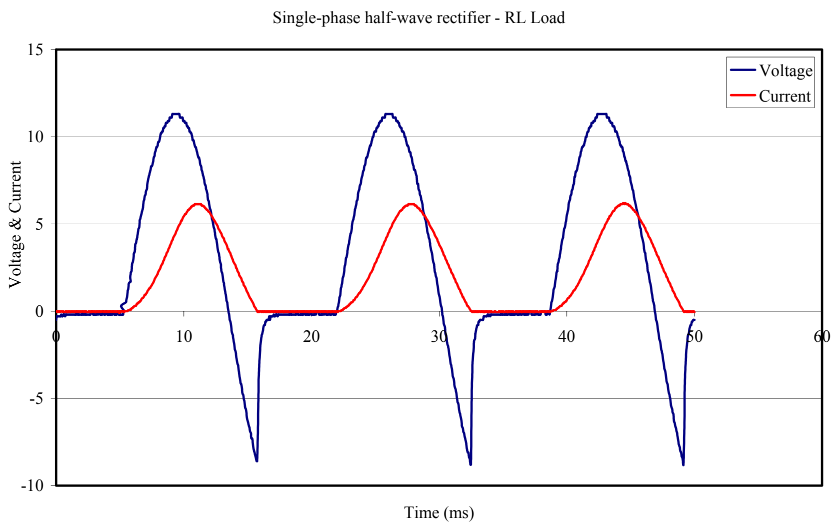


Figure 1-10 Single-phase half-wave rectifier without freewheeling diode

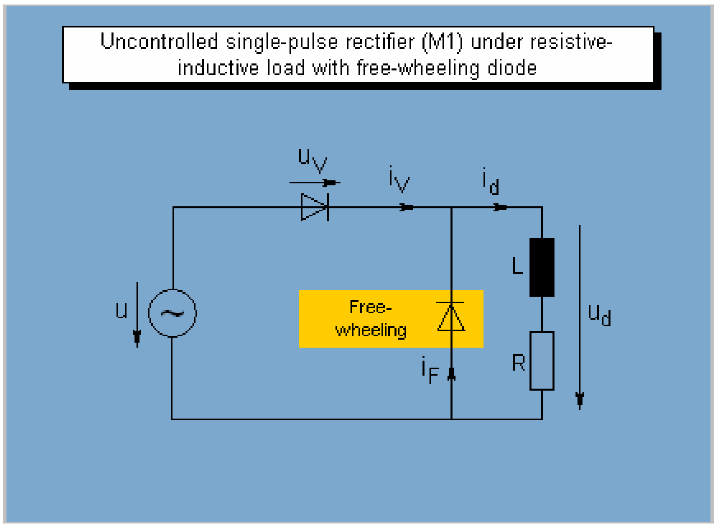


Figure 1-11 Single-phase half-wave rectifier with freewheeling diode

Fig. 1-12 shows the results of the voltage and current output waveforms with freewheeling diode.

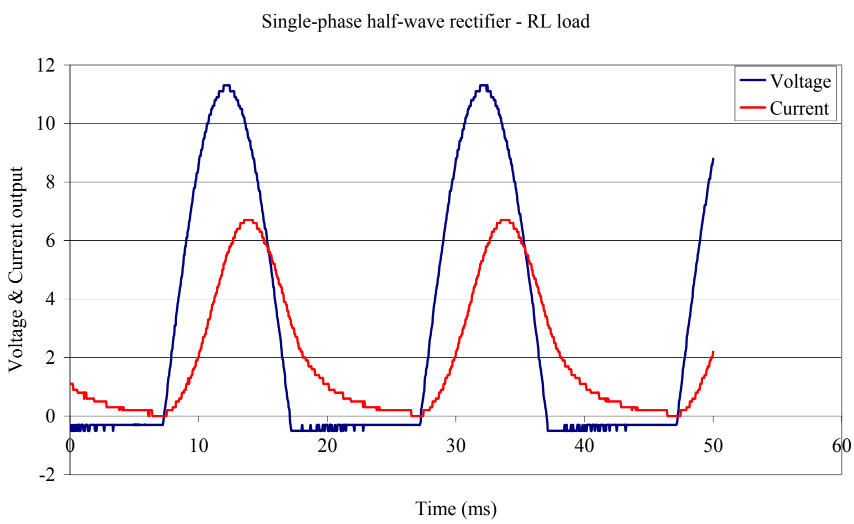


Figure 1-12 Single-phase half-wave rectifier waveform with freewheeling diode

Report:

- Prepare a report that summarizes your results. All plots should be included in the report.
- Shows and records all results from the PSpice simulations.
- Show and records all results from the Experiment.
- Compare the results from the PSpice simulation and the experiment, write your comments.

LAB 2: THREE-PHASE BRIDGE RECTIFIER – R & RL LOADS - PSpice

LARNING OBJECTIVES

After mastering this unit you will:

- Understand the structure & principles of three-phase, full-wave rectifier circuits.
- Understand the difference between three phase and single phase full-wave rectifiers.
- Be able to model a three phase diode rectifier circuit.

REFERENCE: Rashid, *Power Electronics*, Circuits, Devices, & Applications, Third Edition, Pearson – Prentice Hall, 2003.

READING ASIGNMENT

Read section 3.7 & 3.8 in chapter 3 of the text book.

THEORY

The circuit diagram of a three-phase full-wave diode rectifier is shown in Fig. 2-1. The performance parameters of the rectifier with a purely resistive load are given by:

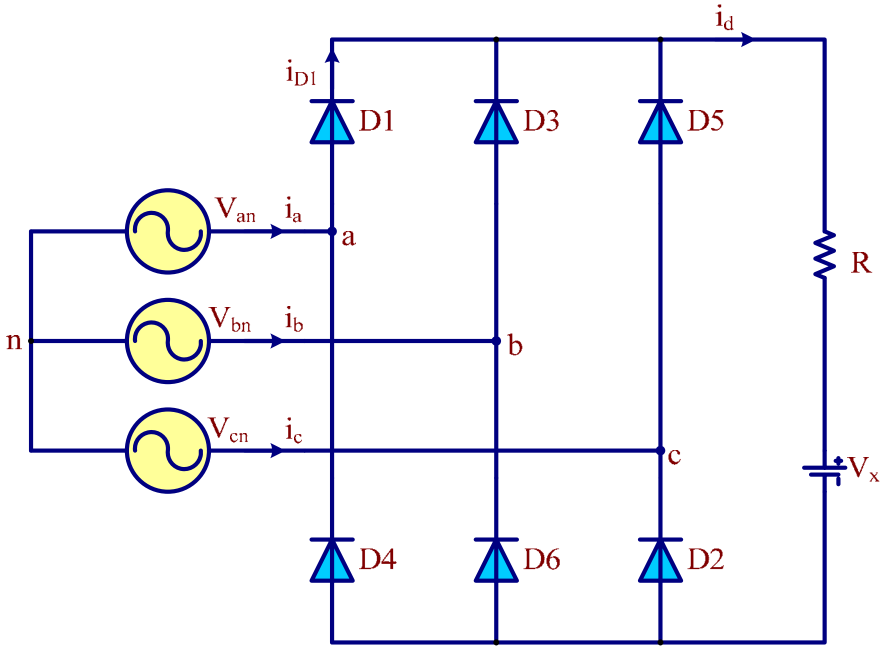


Figure 2-1 Three-phase full-wave rectifier

1. RMS input voltage, V_s.
2. Peak input voltage, $V_m = \sqrt{2}V_s$
3. Supply frequency, f_s.
4. $v_{an} = V_m \sin(\omega t)$; $v_{bn} = V_m \sin(\omega t - 120^o)$; $v_{cn} = V_m \sin(\omega t + 120^o)$
5. $v_{ab} = \sqrt{3}\ V_m \sin(\omega t + 30^o)$; $v_{bc} = \sqrt{3}\ V_m \sin(\omega t - 90^o)$; $v_{ca} = \sqrt{3}\ V_m \sin(\omega t + 90^o)$

6. Average value of the output (load) voltage, $$Vdc = \frac{2}{2\pi/6}\int_0^{\pi/6}\sqrt{3}V_m\cos(\omega t)\cdot d(\omega t) = \frac{3\sqrt{3}}{\pi}V_m = 1.654V_m$$
7. Average value of the output (load) current, $I_{o(dc)} = V_{o(dc)}/R = 1.654\ V_m/R$.
8. Output DC power, $P_{o(dc)} = V_{o(dc)}\ I_{o(dc)}$.
9. RMS value of the output voltage, $V_{o(rms)} = \left(\frac{3}{2}+\frac{9\sqrt{3}}{4\pi}\right)^{1/2} V_m = 1.6554V_m$
10. RMS value of the output current, $I_{o(rms)} = V_{o(rms)}/R = 1.6554\ V_m/R$.
11. RMS value of the diode current, $$I_r = \left[\frac{4}{2\pi}\int_0^{\pi/6} I_m^2\cos^2\omega t\cdot d(\omega t)\right]^{1/2} = I_m\left[\frac{1}{\pi}\left(\frac{\pi}{6}+\frac{1}{2}\sin\frac{2\pi}{6}\right)\right]^{1/2} = 0.5518\cdot I_m$$
12. RMS value of the transformer secondary current, $$I_s = \left[\frac{8}{2\pi}\int_0^{\pi/6} I_m^2\cos^2\omega t\cdot d(\omega t)\right]^{1/2} = I_m\left[\frac{2}{\pi}\left(\frac{\pi}{6}+\frac{1}{2}\sin\frac{2\pi}{6}\right)\right]^{1/2} = 0.7804\cdot I_m$$

The instantaneous output voltage is given by:

$$v_o(t) = 0.9549\cdot V_m\cdot\left(1+\frac{2}{35}\cos(6\omega t)-\frac{2}{143}\cos(12\omega t)+\ldots\right)$$

Where $\omega = 2\pi\ f_s = 2\pi \times 60 = 377$ rad/sec.

PSpice SIMULATION

```
Three-Phase Bridge Rectifier with RL load
VAN   8   0   SIN (0 11.5V     60HZ  0   0   0)
VBN   2   0   SIN (0 11.5V     60HZ  0   0   240DEG)
VCN   3   0   SIN (0 11.5V     60HZ  0   0   240DEG)
L42   6   7   22.3mH
H11   4   6   17
VX    7   5   DC 0V   ; Voltage source to measure the output current
VY    8   1   DC 0V   ; Voltage source to measure the input current
D1    1   4   DMOD
D3    2   4   DMOD
D5    3   4   DMOD
D2    5   3   DMOD
D4    5   1   DMOD
D6    5   2   DMOD
.MODEL  DMOD     D (IS=2.22E-15 BV=1800V) ;  Diode model parameters
.TRAN10US   50MS  33.333MS 10US   ; Transient analysis
.PROBE                      ; Graphics postprocessor
.FOUR     60Hz   I(VY)
.options    ITL5=0    abstol=1.00n   reltol=0.01 vntol=1.00m
.END
```

1. Observe and record the waveforms of the load voltage V_0 and the load current I_0 with input voltage $V_{s(L-L)} = 14.1V$(rms).

2. Measure the average load voltage V_0 (dc), the rms load voltage V_0 (rms), the average load current I_0 (dc), the rms load current I_0 (rms), the rms input current I_a (rms) Phase 1, the rms input voltage for the phase V_{an} and the average load power P_L. (**Tabulate your results**)
3. Repeat 2 with load resistance R (H11) only.

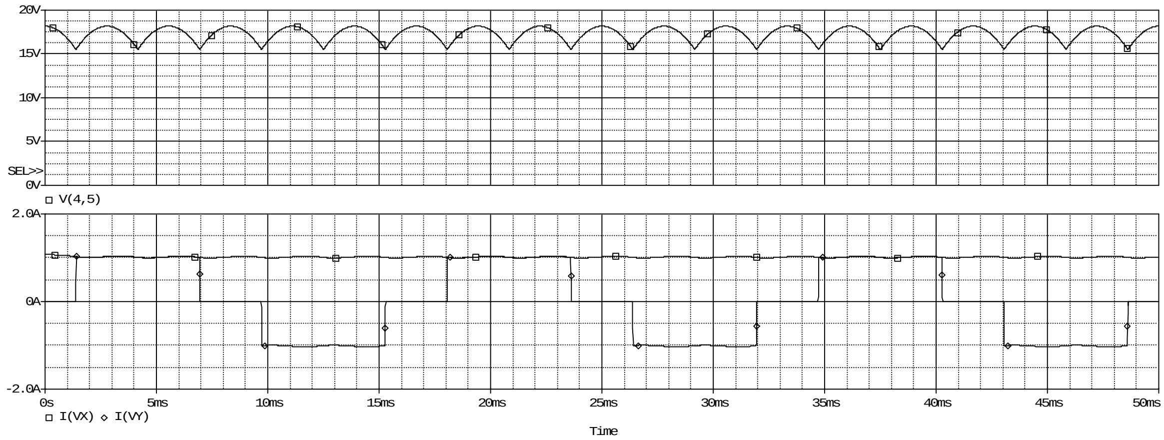


Figure 2-2 Three-phase full-wave rectifier – current & voltage waveforms

Measures Used to Quantify Power Quality

In this lab we will use four values to quantify the nonsinusoidal proprieties of a waveform. The four measures are Total Harmonic Distortion (THD), Crest Factor (CF), Form Factor (FF), and the input Power Factor (PF).

Total Harmonic Distortion

The total harmonic distortion of a waveform is the ratio of the sum of all the harmonic components except the fundamental over the fundamental component. Using a Fourier series, any periodic waveform can be represented by a series of sinusoids. The fundamental component of the series is the predominant frequency, in our case 60 Hz. The remaining components are the magnitudes of the sinusoids at integer multiples of the fundamental frequency.

$$THD = \sqrt{\frac{\sum_{n=1} I_{n,rms}^2}{I_{1,rms}^2}} = \frac{\sqrt{\sum_{n=1} I_{n,rms}^2}}{I_{1,rms}}$$

Crest Factor

The crest factor of a waveform is the ratio of the wave's peak value to its rms value. This measure is useful when there is significant ripple in the sinusoidal waveform.

$$Crest\ Factor = \frac{I_{peak}}{I_{rms}}$$

Form Factor

The form factor of a waveform is the ratio of the wave's rms value to its average value. This measure is useful when compare two quasi-dc waveforms.

$$Form\ Factor = \frac{I_{rms}}{I_{avg}}$$

Assignment

1. Construct a full-wave rectifier and a six-pulse bridge in PSpice. Perform a transient analysis of each circuit for three complete 60 Hz periods.
   a. Plot the source current(s) using Probe.
   b. Determine the harmonic spectrum of each current using the FFT function of Probe.
   c. Create a table of the first 25 harmonic values obtained in step b.
   d. Determine the THD, Form Factor, Power Factor, and Crest Factor for the source currents in each circuit.

Report:

1. Present all recorded waveforms and discuss all significant points.
2. Tabulate all results generated by the Pspice simulation and calculation.
3. Write your comments.

LAB 3: THREE-PHASE BRIDGE RECTIFIER - R & RL LOADS - EXPERIMENT

LARNING OBJECTIVES

After mastering this unit you will:

- Observe and verify the three-phase, full-wave rectifier circuits.
- Understand the difference between three phase and single phase full-wave rectifiers.
- Be able to model a three phase diode rectifier circuit.

REFERENCE: Rashid, *Power Electronics*, Circuits, Devices, & Applications, Third Edition, Pearson – Prentice Hall, 2003.

EXPERIMENT

Three-phase bridge rectifier under resistive and inductive load.

Application:

A three-phase bridge rectifier is used as an input stage in many applications (e.g. variable speed ac motor) devices.

Apparatus:

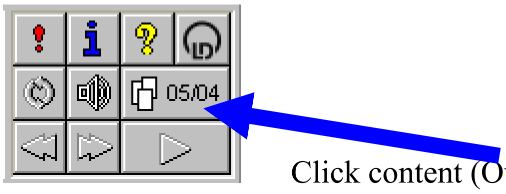
Com3LAB board is equipped with phase commutated converter, load resistance, load inductance, load capacitance, multimeter, dual oscilloscope, and function generator. You will use from the phase commutated converter the six diodes V21, V23, V25, V24, V26, & V22.

- Six diode with ratings of at least 1A & 50V.
- An R load R_L (H11) = 15Ω.
- Oscilloscope has two channels Y1, Y2, & COM.
- AC & DC voltmeters & ammeters & one nonconductive shunt.

Experimental procedure:

1. Set up the circuit as shown in Figure 1.
2. Connect the measuring instruments as required.
3. Observe and record the waveforms of the load voltage V_0 and the load current I_0 with input voltage $V_{s(L-L)}$ = 14.10V(rms).
4. Measure the average load voltage V_0 (dc), the rms load voltage V_0 (rms), the average load current I_0 (dc), the rms load current I_0 (rms), the rms input current I_a (rms) Phase a, the rms input voltage for the phase V_{an} and the average load power P_L.

After completing the connection of the above circuit you can run the experiment as the following:
Turn **ON** the board.
Click → **Start button** → **Programs** → **COM3LAB** → **70021 power electronic I**.
Window will open, type your initial name (such as "**mk**") and then click **Continue**.
Click again Continue.



Click content (Overview/Exist) to select the experiment.
Select from the uncontrolled rectifier three phase circuit as shown in Figure 3-1.

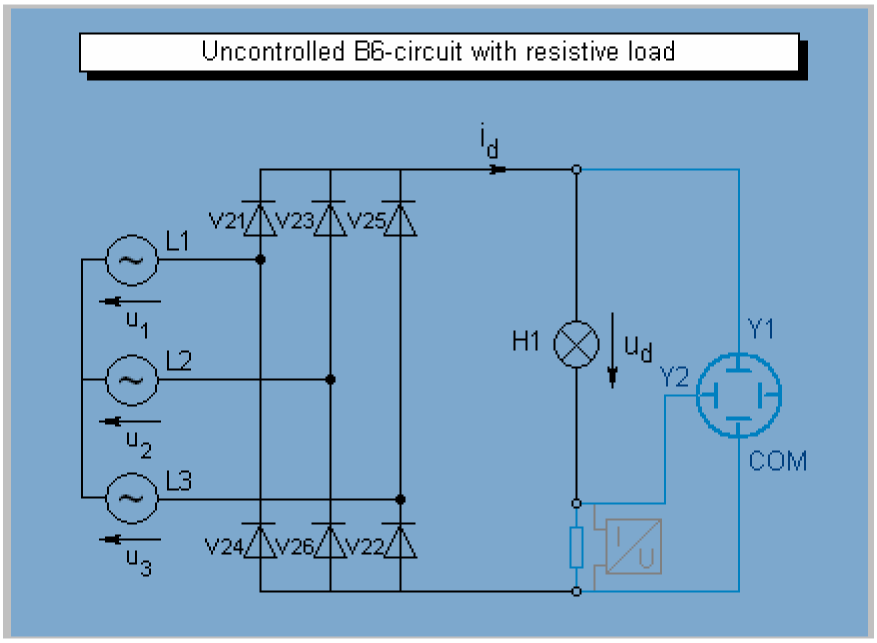


Figure 3-1 Three-Phase Bridge Rectifiers

The oscilloscope will show you the output voltage and load current as shown in Fig. 3-2.

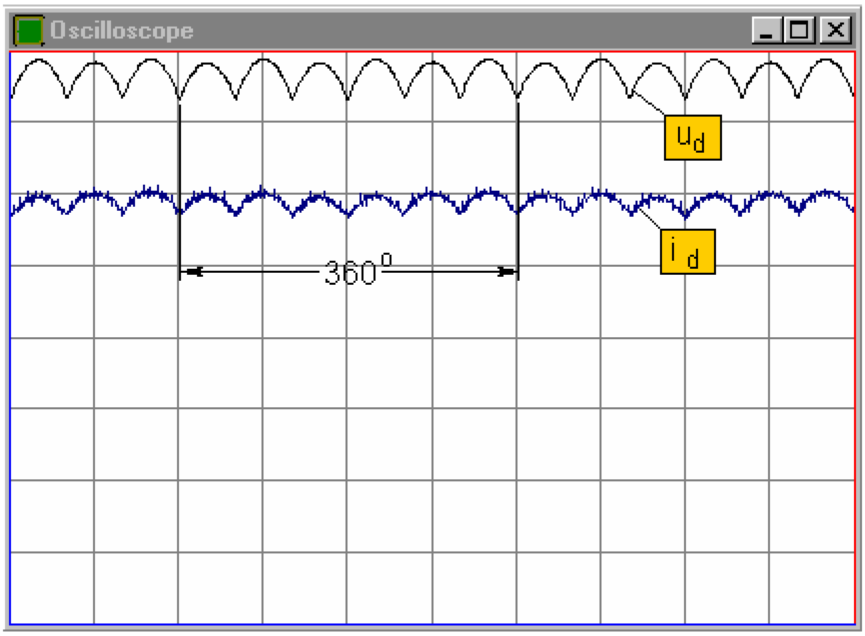


Figure 3-2 Three-Phase Bridge Rectifiers, voltage & current output for resistive load

Repeat the experiment by using inductive load with adding the inductor L11 between the H11 and the shunt resistance as shown in Fig. 3-3.

Fig. 3-4 shows the waveforms of the three-phase bridge rectifier with inductive load.

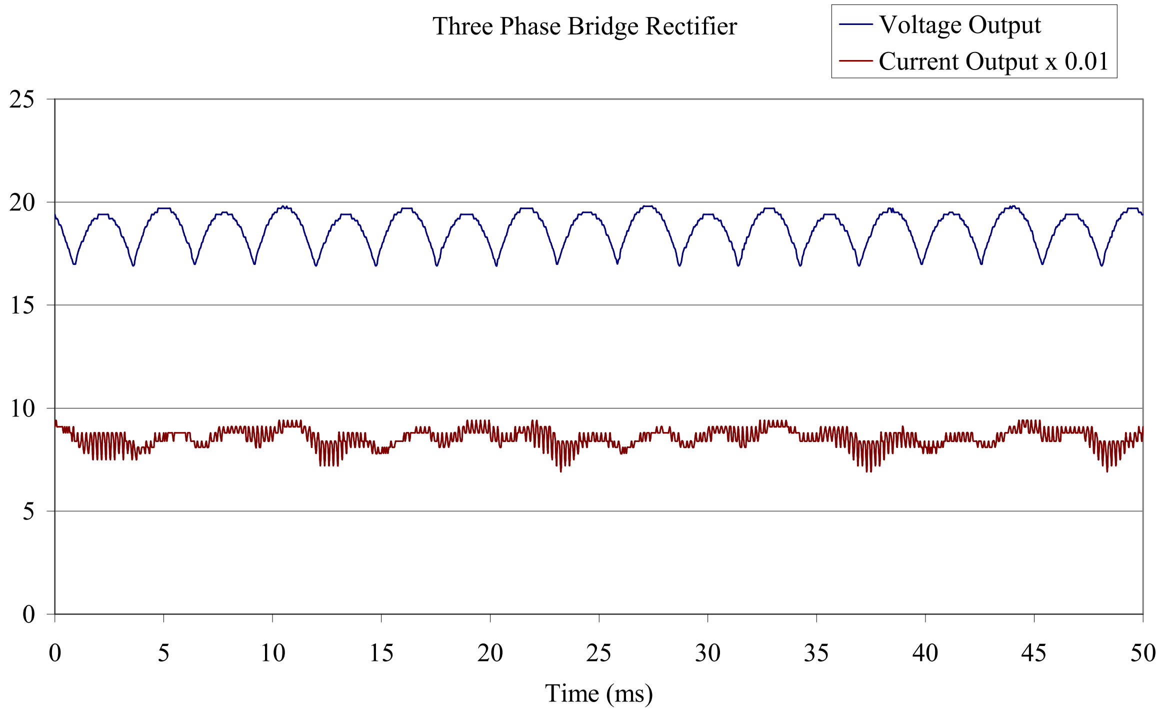


Figure 3.4 Three-Phase Bridge Rectifiers, voltage & current output for inductive load

Report:

4. Present all recorded waveforms and discuss all significant points.
5. Compare all results generated by the Pspice simulation and with the experimental and write your comments.

LAB 4: SINGLE-PHASE BRIDGE CONTROLLED RECTIFIER - EXPERIMENT

LEARNING OBJECTIVES

After mastering this unit you will:

- Know the operation of a single-phase controlled rectifier with both inductive and resistive loads.
- Know the effects' of delay angle (control variable) on the performance of the rectifier
- Know about the gating requirements of thyristor controlled rectifiers.

READING ASSIGNMENT

Read sections 10-3 in chapter 10 of the textbook. Read a textbook on power electronics on single-phase full converters.

THEORY

In this laboratory, we will construct a single-phase controlled rectifier to give a variable DC output voltage. This is shown in Fig. 4-1. The gating signals must be isolated by either pulse transformers or optocoupler.

Resistive Load

The average output voltage of the rectifiers is given by

$$V_{dc} = \frac{2}{2\pi}\int_{\alpha}^{\pi} V_m \cdot \sin(\omega t)\cdot d(\omega t) = \frac{V_m}{\pi}\left[-\cos(\omega t)\right]_{\alpha}^{\pi} = \frac{V_m}{\pi}(1+\cos\alpha)$$

Where: V_m is the peak value of the input supply voltage.

The rms value of the output voltage is given by:

$$V_{rms} = \left[\frac{2}{2\pi}\int_{\alpha}^{\pi} V_m^2 \cdot \sin^2 \omega t \cdot d(\omega t)\right]^{1/2} = \left[\frac{V_m^2}{2\pi}\int_{\alpha}^{\pi}(1-\cos 2\omega t)\cdot d(\omega t)\right]^{1/2}$$

$$= \left[\frac{V_m^2}{2\pi}\left(\omega t - \frac{\sin(2\omega t)}{2}\right)\Bigg|_{\alpha}^{\pi}\right]^{1/2} = \frac{V_m}{\sqrt{2}}\left[\frac{1}{\pi}\left(\pi - \alpha + \frac{\sin 2\alpha}{2}\right)\right]^{1/2}$$

Inductive Load

The average output voltage of the rectifiers is given by

$$V_{dc} = \frac{2}{2\pi}\int_{\alpha}^{\alpha+\pi} V_m \cdot \sin(\omega t)\cdot d(\omega t) = \frac{2V_m}{2\pi}\left[-\cos(\omega t)\right]_{\alpha}^{\alpha+\pi} = \frac{2V_m}{\pi}\cos\alpha$$

Where: V_m is the peak value of the input supply voltage.

The rms value of the output voltage is given by:

$$V_{rms} = \left[\frac{2}{2\pi}\int_{\alpha}^{\alpha+\pi} V_m^2 \cdot \sin^2 \omega t \cdot d(\omega t)\right]^{1/2} = \frac{V_m}{\sqrt{2}} = V_s$$

Form Factor = $\frac{V_{o,rms}}{V_{o,dc}}$ and Ripple Factor = $\sqrt{FF^2 - 1}$

You will investigate the controlled bridge circuits under different types of loads. First, the controlled two-pulse bridge circuit is investigated under a purely resistive load as shown in Fig. 4-1.

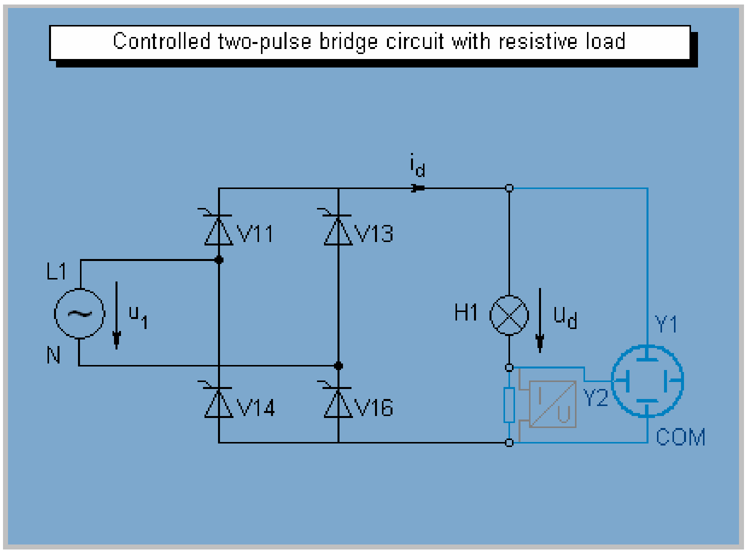


Figure 4-1 Single-phase controlled bridge rectifier with purely resistive load

Setup the circuit shown in Fig. 4-1 and set the control unit to the B2 operating mode. Start determining the DC voltage U_d and DC current i_d for $\alpha = 0$ to $\alpha = 180$ degrees.

1. Observe and record the waveforms of the load voltage V_0 and the load current I_0 with input voltage $V_{s(L-L)} = 14.1$V(rms).
2. Measure the average load voltage V_0 (dc), the rms load voltage V_0 (rms), the average load current I_0 (dc), the rms load current I_0 (rms), the rms input current I_a (rms) Phase 1, the rms input voltage for the phase V_{an} and the average load power P_L. (**Tabulate your results**)
3. Repeat 2 with adding inductance L11 as shown in Fig. 4-2..

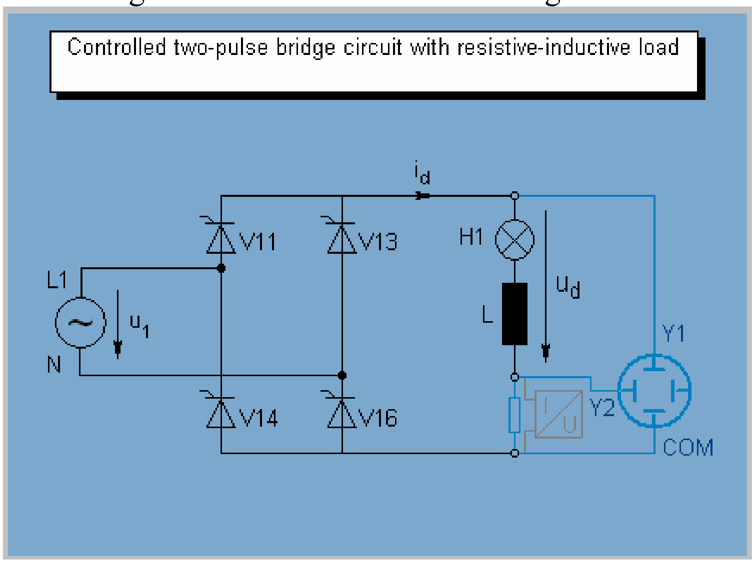


Figure 4-2 Single-phase controlled bridge rectifier with inductive load

Determining the control characteristic:

In the following the control characteristic $U_{d\alpha}(\alpha)$ of B2C circuit are determined for resistive load. To do this the trigger delay angle α is varied in the range between 0..180 degrees. Click on the converter analyzer as shown in Fig. 4-3.

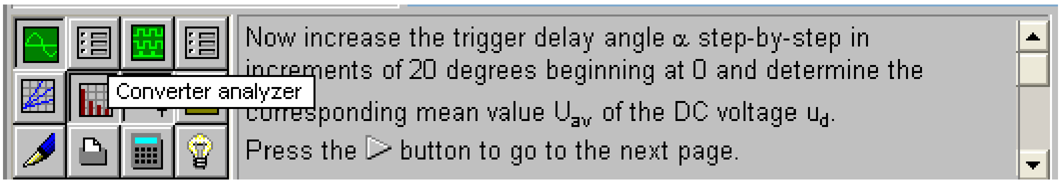


Figure 4-3 Converter analyzer

Now increase the trigger delay angle α step-by-step in increments of 20 degrees beginning at 0 and determine the corresponding mean value U_{av} of the DC voltage u_d. Each reading will be interred manually by typing the trigger delay α and the mean value U_{av} in the **Characteristic Curve Plotter** as shown in Fig. 4-4. After the completing the table, click on **Show** and you will see the plot.

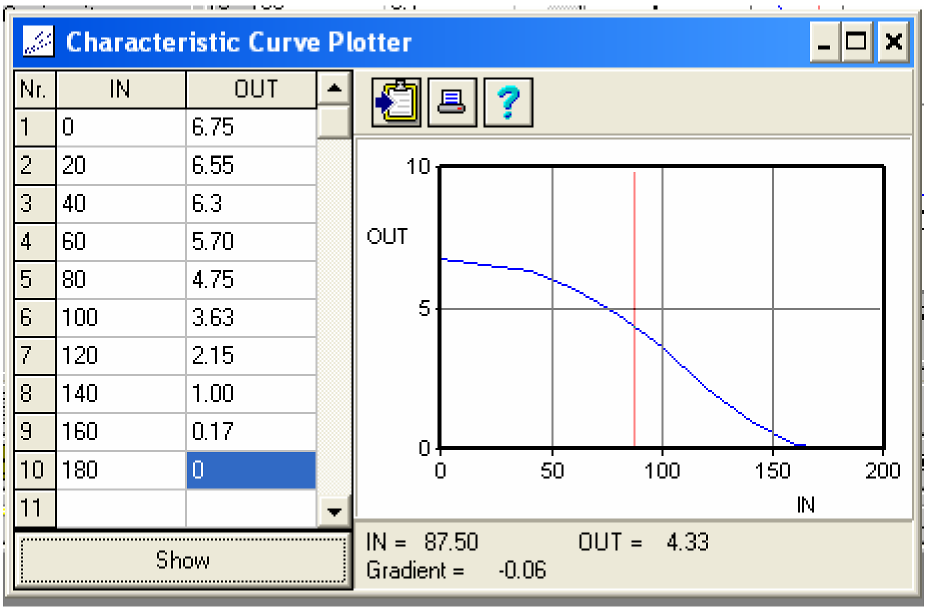


Figure 4-4 Characteristic Curve Plotter

Determine the mean value and the RMS value of the DC voltage U_d and from these values calculate the form factor and the ripple.

Repeat the control characteristic procedure for inductive load (H11+L11).

Report:

1. Present all recorded waveforms and discuss all significant points.
2. Tabulate the results of the experiment, write your comments.
3. What are the main differences between resistive and inductive load?

LAB 5: THREE-PHASE CONTROLLED BRIDGE RECTIFIER - PSpice

LARNING OBJECTIVES

After mastering this unit you will:

- Understand the structure & principles of three-phase, full-wave rectifier circuits.
- Understand the difference between three phase and single phase full-wave rectifiers.
- Be able to model a three phase diode rectifier circuit.

REFERENCE: Rashid, *Power Electronics*, Circuits, Devices, & Applications, Third Edition, Pearson – Prentice Hall, 2003.

READING ASIGNMENT

Read section 10.6 in chapter 10 of the text book.

THEORY

The circuit diagram of a three-phase full-wave thyristors rectifier is shown in Fig. **5**-1. The performance parameters of the rectifier with a purely resistive load are given by:

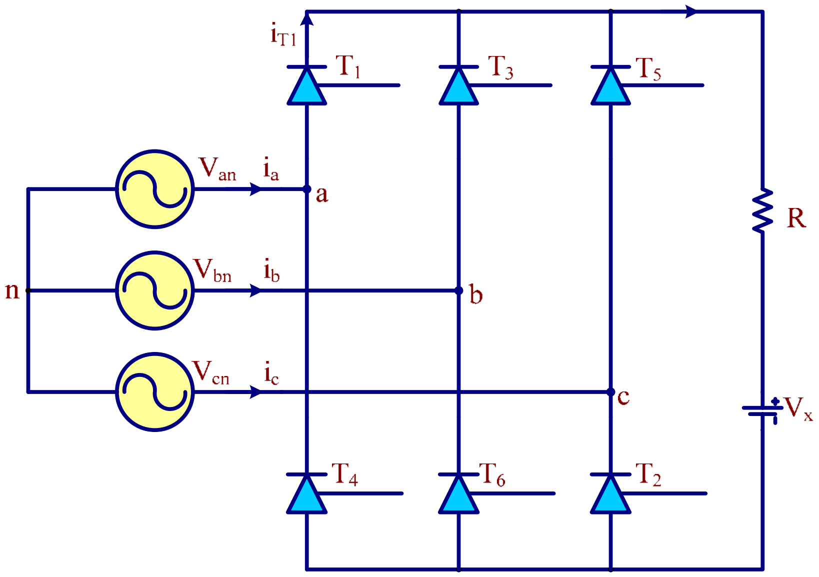


Figure 5-1 Three-phase full-wave rectifier

MATHEMATICAL ANALYSIS

Analysis of this three-phase controlled rectifier is in many ways similar to the analysis of single-phase bridge rectifier circuit. We are interested in output voltage and the source current. The average output voltage, the rms output voltage, the ripple content in output voltage, the total rms line current, the fundamental rms current, THD in line current, the displacement power factor and the apparent power factor are to be determined. In this section, the analysis is carried out assuming that the load current is a steady dc value.

AVERAGE OUTPUT VOLTAGE

Before getting an expression for the output voltage, it is preferable to find out how the output voltage waveform varies as the firing angle is varied. In one cycle of source voltage, six pairs conduct, each pair for 60^o. This means that the period for output waveform is one-sixth of the period of line voltage. The output waveform repeats itself six times in one cycle of input voltage. The waveform of output voltage can be determined by considering one pair. It is seen that when $v_a(t) = V_m * \text{Sin}(\omega t)$, SCR T1 and T6 conduct when ωt varies from $30^o + \alpha$ to $90^o + \alpha$, where a is the firing angle. Then

$$v_o = \sqrt{3}V_m \cdot \sin(\omega t + 30^o), \qquad for \quad (\alpha + 30^o) < \omega t < (\alpha + 90^o)$$

The waveform of output can be plotted for different firing angles. The applet below takes in the firing angle as an input and plots the output. The peak line-to-line voltage is marked as 'U' and the applet starts with the instant an SCR is fired and displays the output waveform for one input cycle period.

The average output voltage of the bridge circuit is calculated as follows, with a change in variable, where $\omega t = \alpha + 60^o$.

$$V_{0,avg}(\alpha) = \frac{\sqrt{3}V_m}{\pi/3} \int_{\alpha+\pi/3}^{\alpha+(2\pi/3)} \sin(\omega t) \cdot d(\omega t) = \frac{3\sqrt{3}}{\pi} V_m \cdot \cos(\alpha)$$

In the expression above, V_m is the amplitude of phase voltage of 3-phase supply.

RMS OUTPUT VOLTAGE

The rms output voltage is calculated as follows:

$$V_{o,rms}(\alpha) = \sqrt{3}V_m \cdot \sqrt{\frac{3}{\pi} \int_0^{\pi/3} \sin^2(\omega t + \frac{\pi}{3} + \alpha) \cdot d(\omega t)} = \sqrt{3}V_m \cdot \sqrt{\frac{1}{2} + \frac{3\sqrt{3}}{4\pi} \cos(2\alpha)}$$

The ripple factor of the output voltage is then:

$$RF(\alpha) = \frac{\sqrt{(V_{o,rms}(\alpha))^2 - (V_{o,avg}(\alpha))^2}}{V_{o,avg}(0^o)}$$

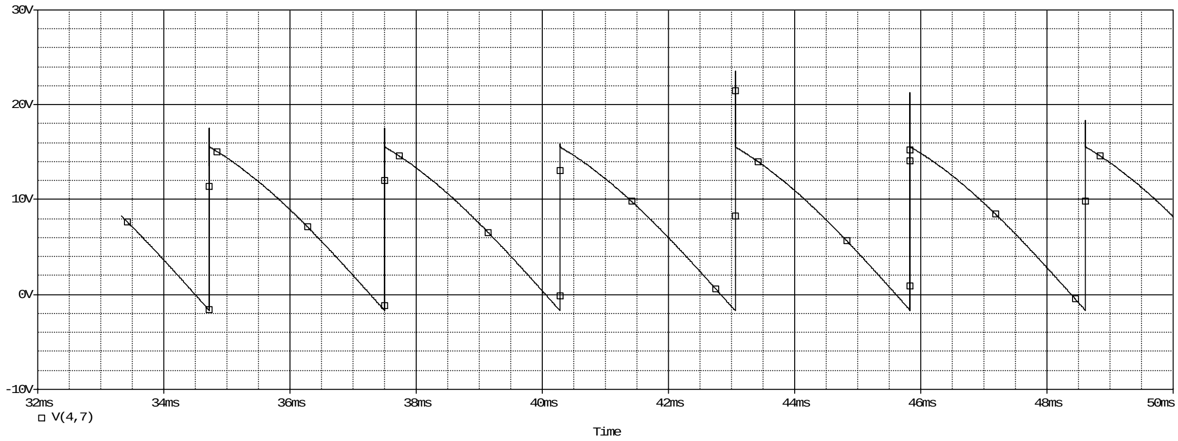
PSPICE SIMULATION

The Pspice program for simulation of this three-phase rectifier circuit is presented below. The model used for the SCRs is the same as defined for the single-phase fully-controlled bridge rectifier. The three-phase bridge rectifier contains six SCRs and it is necessary to define six pulse sources, one for each SCR. The pulse sources have been defined for a firing angle of 30^o and the frequency of the three-phase source is 60 Hz. At any time two SCRs need to conduct, one from the top half and another bottom half and hence only if one SCR is triggered at a time, conduction may never get established. To overcome this problem, two SCRs are triggered at the same time. For example, when SCR T_2 is to be triggered, SCR T_1 is also triggered. In the same way, when SCR T_3 is to be triggered, SCR T_2 is also triggered and so on. In order to affect this in program, one voltage-controlled

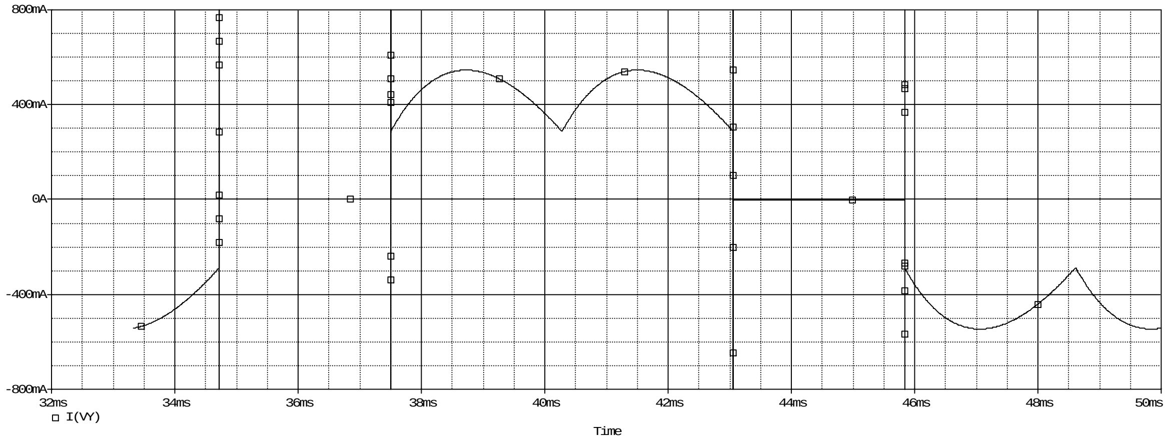
voltage source is defined for each SCR. The dependent source defined for SCR T_1 is dependent on two sources, the pulse source that defines when SCR T_1 is to be triggered and the pulse source that defines when SCR T_2 is to be triggered.

```
* Three-Phase Full-Wave Fully-Controlled Bridge Rectifier
Van    14    0     SIN(0  11.5V  60HZ)
Vbn    2     0     SIN(0  11.5V  60HZ  0  0  -120DEG)
Vcn    3     0     SIN(0  11.5V  60HZ  0  0  -240DEG)
Vg1    8     4     PULSE (0V 10V 4166.7US 1NS 1NS 100US 16666.7US)
Vg3    9     4     PULSE (0V 10V 9722.2US 1NS 1NS 100US 16666.7US)
Vg5    10    4     PULSE (0V 10V 15277.8US 1NS 1NS 100US 16666.7US)
Vg2    11    3     PULSE (0V 10V 6944.4US 1NS 1NS 100US 16666.7US)
Vg4    12    1     PULSE (0V 10V 12500.0US 1NS 1NS 100US 16666.7US)
Vg6    13    2     PULSE (0V 10V 1388.9US 1NS 1NS 100US 16666.7US)
R      4     6     17
L      6     7     22.3MH
VX     7     5     DC   0V  ; Load battery voltage
VY     14    1     DC   0V  ; Voltage source to measure supply current
*  Subcircuit calls for thyristor model
XT1  1  4  8   4  SCR   ; Thyristor T1
XT3  2  4  9   4  SCR   ; Thyristor T3
XT5  3  4  10  4  SCR   ; Thyristor T5
XT2  5  3  11  3  SCR   ; Thyristor T2
XT4  5  1  12  1  SCR   ; Thyristor T4
XT6  5  2  13  2  SCR   ; Thyristor T6
*  Subcircuit for switched thyristor model
.SUBCKT   SCR   1     2       3        2
*     model  anode  cathode  +control  -control
*     name                   voltage   voltage
S1  1  5  6  2  SMOD      ; Switch
RG  3  4  50
VX  4  2  DC  0V
VY  5  7  DC  0V
DT  7  2  DMOD            ; Switch diode
RT  2  6  1
CT  6  2  10UF
F1  2  6  POLY(2)  VX  VY  0  50  11
.MODEL  SMOD  VSWITCH (RON=0.0125 ROFF=10E+5 VON=0.5V VOFF=0V)
.MODEL  DMOD  D(IS=2.2E-15 BV=1200V TT=0)  ; Diode model parameters
.ENDS  SCR                ; Ends subcircuit definition
.TRAN  50US  50MS  33.33MS  50US   ; Transient analysis
.PROBE                    ; Graphics post-processor
.FOUR  60HZ  I(VY)
.OPTIONS  ABSTOL = 1.00N RELTOL = 0.01 VNTOL = 0.01 ITL5=0 ; convergence
.END
```

The output voltage waveform



The line current (phase A) waveform



The voltage (SCR T_1) waveform

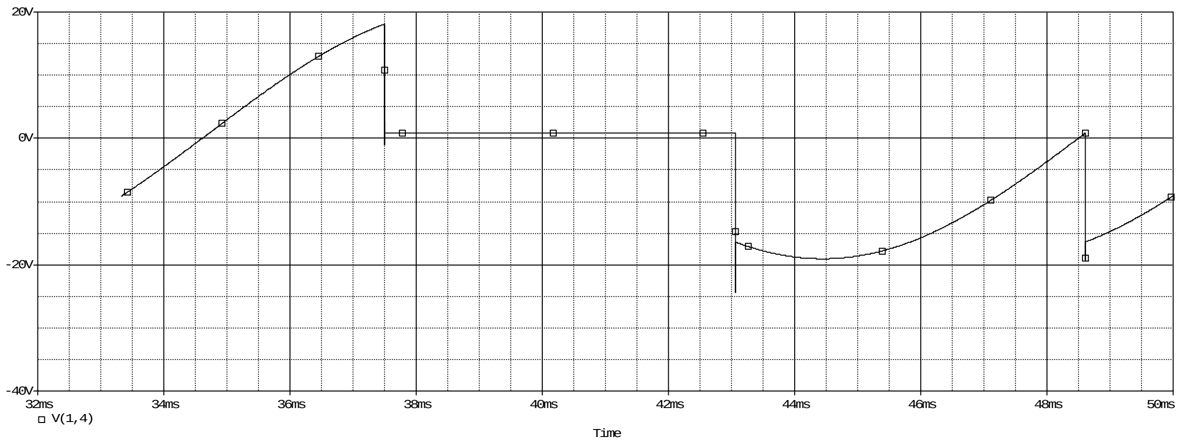


Figure 5-2 Three-phase controlled rectifier – output voltage, line current, & voltage across T_1

Measure the average load voltage V_0 (dc), the rms load voltage V_0 (rms), the average load current I_0 (dc), the rms load current I_0 (rms), the rms input current I_a (rms) Phase 1, the rms input voltage for the phase V_{an} and the average load power P_L. (**Tabulate your results**)

Repeat 2 with load resistance H11 only.

Assignment

Construct a full-wave rectifier and a six-pulse bridge in PSpice. Perform a transient analysis of each circuit for three complete 60 Hz periods.

a. Plot the source current(s) using Probe.
b. Determine the harmonic spectrum of each current using the FFT function of Probe.
c. Create a table of the first 25 harmonic values obtained in step b.
d. Determine the THD, Form Factor, Power Factor, and Crest Factor for the source currents in each circuit.

Report:

1. Present all recorded waveforms and discuss all significant points.
2. Tabulate all results generated by the Pspice simulation and calculation.
3. Write your comments.

LAB 6: THREE-PHASE CONTROLLED BRIDGE RECTIFIER – EXPERIMENT

LARNING OBJECTIVES

After mastering this unit you will:

- Understand the structure & principles of three-phase, full-wave rectifier circuits.
- Understand the difference between three phase and single phase full-wave rectifiers.
- Be able to model a three phase diode rectifier circuit.

REFERENCE: Rashid, *Power Electronics*, Circuits, Devices, & Applications, Third Edition, Pearson – Prentice Hall, 2003.

READING ASIGNMENT

Read section 10.6 in chapter 10 of the text book.

Controlled line-commutated converters – The COM3LAB control unit

The Com3Lab control unit performs all the operations needed to control the external and self-commutated converter circuits found on the board. The trigger delay angle α can continuously set between 0 to 180 degrees.

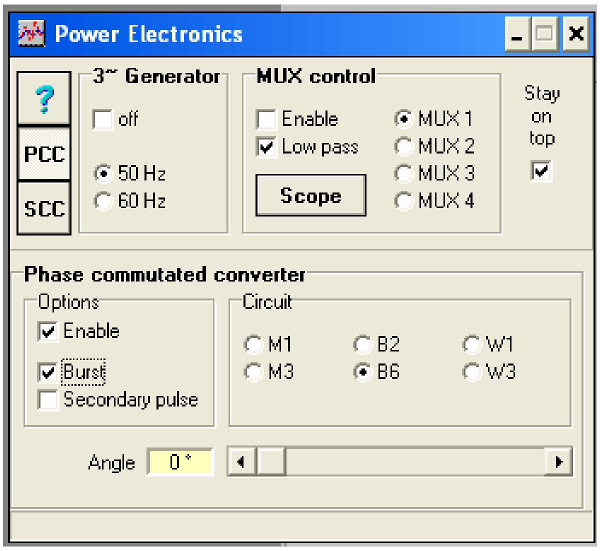


Figure 6-1 The COM3LAB control unit

Build the circuit as is shown in Figure 6-2. You shall now record and analyze the curve of the DC voltage and current for several different settings of the trigger delay angleα.

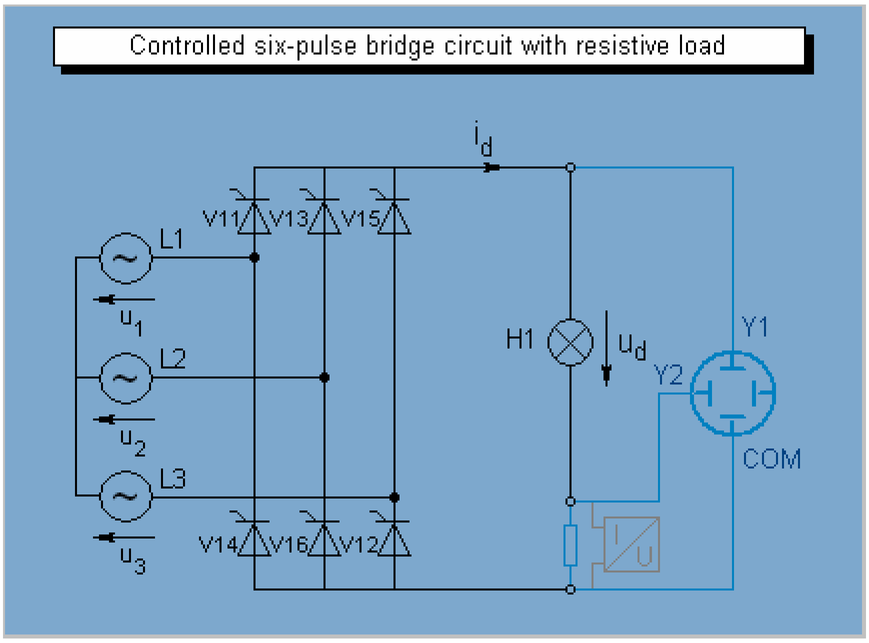


Figure 6-2 Three-Phase Bridge Controlled Rectifier

Result:
For trigger delay angle α > 0 a nearly sawtooth-shaped DC voltage results which becomes intermittent as of the trigger delay angle of 60 degrees. Qualitatively speaking the DC current has the same curve. At a trigger delay angle of 120 degrees both quantities become zero. Figure 6-3 shows the voltage and the current output for the delay angle α = 30 degrees.

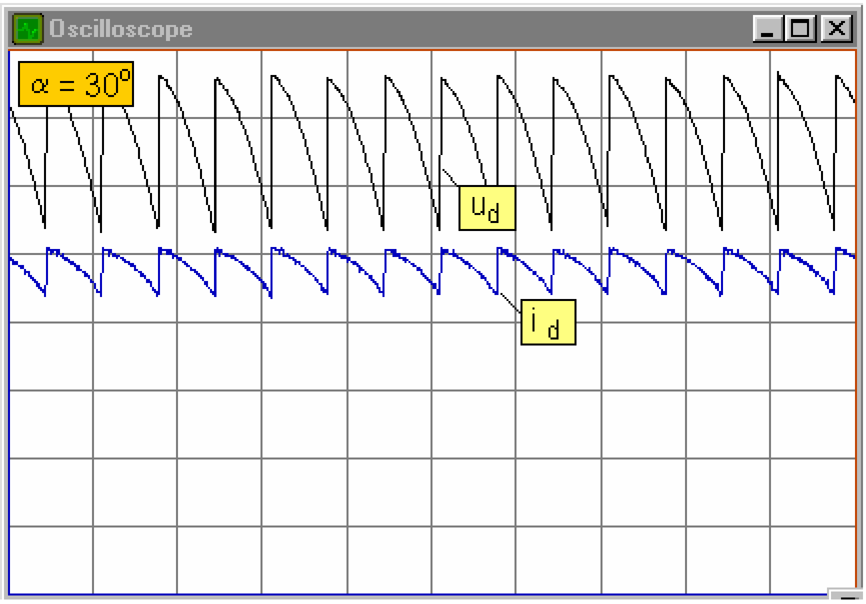


Figure 6-3 Three-Phase Bridge Controlled Rectifier for $\alpha = 30^o$

Control Characteristic

In the concluding experiment, we shall determine the control characteristic of the B6C circuit with resistive load.

Setup the original circuit as shown in Fig. 6-2 again and gradually increases the trigger delay angle α in increment of 20 degrees beginning of 0^o and determine the corresponding mean value U_{av} of the DC voltage U_d.

Under resistive load the B6C circuit can be continuously controlled in the range of 0 to 120 degrees. Qualitatively speaking the control characteristic is similar to single-phase controlled bridge rectifier. Fig. 6-4 shows the control characteristic of three phase controlled bridge rectifier.

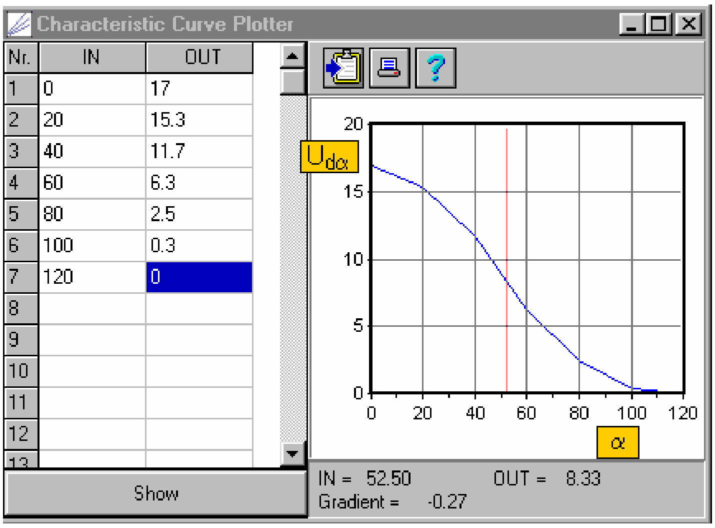


Figure 6-4 Controlled characteristic of three-phase controlled bridge rectifier

Repeat the experiment by using inductive load, H11 and L11, as shown in Fig. 6-2.

Assignment

Construct a full-wave controlled bridge rectifier and perform a transient analysis of each circuit for three complete 60 Hz periods.

a. Plot the source current(s) using Probe.
b. Determine the mean value and the RMS value of the DC voltage U_d.
c. Determine the harmonic spectrum of each current using the FFT function of Probe.
d. Create a table of the first 25 harmonic values obtained in step b.
e. Determine the THD, Form Factor, Power Factor, and Crest Factor for the source currents in each circuit.

<u>Report:</u>

1. Present all recorded waveforms and tables and discuss all significant points.
2. Compare the results of your PSpice simulation and the experiment, write your comments.
3. What are the main differences between resistive and inductive load?

LAB 7: SINGLE-PHASE AC VOLTAGE CONTROLLER – Pspice

LARNING OBJECTIVES

After mastering this unit you will:

- Know the operation of AC voltage controller and the performance parameters.
- Know the techniques of generating the control signals for ac voltage controllers.
- Be able to model and simulate ac voltage controllers.

READING ASIGNMENT

Read section 11.4, 11.5, and 11.12 in chapter 11 of the text book.

THEORY

R - LOAD

The circuit diagram of a single-phase ac voltage controller shown in Fig. 7-1. The performance parameters of an AC voltage controller are given by:

- Supply frequency, f_s.
- RMS input voltage, V_s.
- Load resistance, R.
- The RMS output (load) voltage is given by: $V_{o(rms)} = V_s\left[\frac{1}{\pi}\left(\pi - \alpha + \frac{\sin 2\alpha}{2}\right)\right]^{1/2}$.
- The RMS output (load) current, $I_{o,(rms)} = V_{o(rms)} / R$.
- Output power, $P_o = RI^2_{o(rms)}$.

RL - LOAD

- The current in T1 is given by: $i_1 = \frac{\sqrt{2}Vs}{Z}\left[\sin(\omega t - \theta) - \sin(\alpha - \theta)e^{(R/L)(\alpha/\omega - t)}\right]$. Where α is the delay (firing) angle and θ is the load angle $\theta = \tan^{-1}(\omega L / R)$.
- The extinction angle, β, can be determined from the transcendental equation: $\sin(\beta - \theta) = \sin(\alpha - \theta)\cdot e^{(R/L)(\alpha - \beta)/\omega}$.
- The conduction angle is defined as: δ = β-α.
- The RMS output voltage: $V_{o(rms)} = V_s\left[\frac{1}{\pi}\left(\beta - \alpha + \frac{\sin 2\alpha}{2} - \frac{\sin 2\beta}{2}\right)\right]^{1/2}$.
- The RMS thyristor current I_R: $I_R = \frac{V_s}{Z}\left[\frac{1}{\pi}\int_{\alpha}^{\beta}\left[\sin(\omega t - \theta) - \sin(\alpha - \theta)e^{(R/L)(\alpha/\omega - t)}\right]\cdot d(\omega t)\right]^{1/2}$.
- The RMS output current $I_{o(rms)}$: $I_{o(rms)} = \sqrt{2}I_R$

- The average value of thyristor current: $i_1 = \dfrac{\sqrt{2}Vs}{2\pi \cdot Z}\int_{\alpha}^{\beta}\left[\sin(\omega t-\theta)-\sin(\alpha-\theta)e^{(R/L)(\alpha/\omega - t)}\right]\cdot d(\omega t)$.

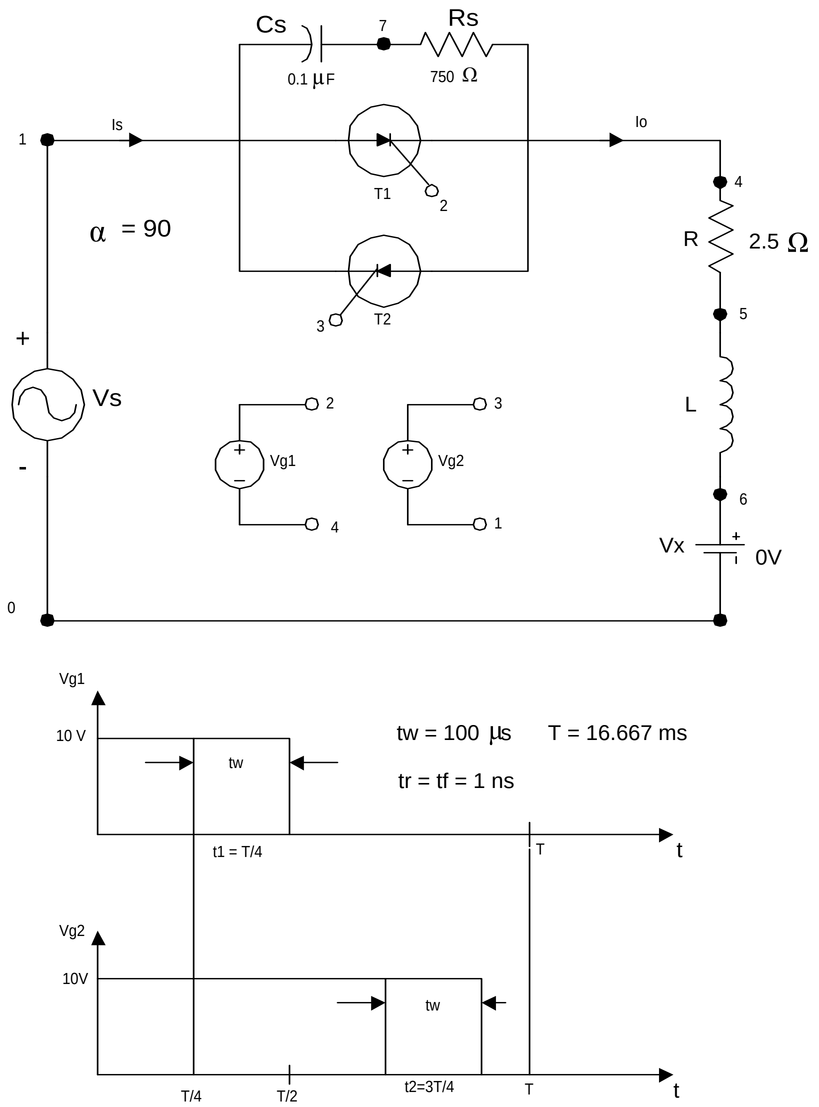


Figure 7-1 Single-phase ac voltage controller circuit with control signal

Resistive Load R(H11)

Go to the menu of Pspice. Choose new and give a file-name. Type the following statement:

Type your name, ID #, Experiment #, Course #, and Date
VS 1 0 SIN (0 11.5V 60HZ)
V_{g1} 2 4 PULSE (0V 10V 4166.7US 1NS 1NS 100US 16666.7US)
V_{g2} 3 1 PULSE (0V 10V 12500.0US 1NS 1NS 100US 16666.7US
R 4 6 15
*L 5 6 313MH
VX6 0 DC 0V ; Voltage source to measure the load current
CS 1 7 0.1UF

```
RS 7    4    750
* Subcircuit call for thyristor model
XT1     1    4    2    4    ASCR ; Thyristor T1
XT2     4    1    3    1    ASCR ; Thyristor T2
* Subcircuit for switched Thyristor model
. SUBCKT ASCR    1  2    3    2
* model     anode  cathode    +control    -control
* name      voltage voltage
S1  1    5    6    2    SMOD      ; Switch
RG 3    4    50
VX 4    2    DC 0V
VY 5    2    DC 0V
RT 2    6    1
CT 6    2    10UF
F1  2    6    POLY (2)  VX VY 0    50  11
. MODEL  SMOD  VSWITCH (RON=0.0125  ROFF=10E+5V VON=0.5V VOFF=0V)
. ENDS     ASCR       ; Ends Subcircuit definition
. TRAN     10US   50MS  0    10US   ; Transient analysis
. PROBE                             ; Graphics post-processor
. options  abstol=1.00n  retol=1.0m  vntol=1.0m  ITL5=0  ;  convergence
. FOUR 60HZ I(VX)  V(4)             ;  Fourier analysis
. END
```

Save the file.
Run analysis.
Plot load current I (VX) and load voltage V(4).

RESULTS

The rms output voltage will depend upon the delay angle or delay time (firing angle). Repeat simulation for the three values of delay angles and complete Table 7-1.

Table 7-1 Resistive Load

Delay angle α	$V_{o(rms)}$ RMS(V(4))	$I_{0(rms)}$ RMS(I(VX))	$I_{T(rms)}$	$I_{T(dc)}$	THD_i	PF
30^o						
60^o						
90^o						
120^o						

Repeat the PSpice simulation for RL-load (H11 & L11) and record all results in Table 7-2.

Table 7-2 Inductive Load

Delay angle α	$V_{o(rms)}$ RMS(V(4))	$I_{0(rms)}$ RMS(I(VX))	$I_{T(rms)}$	$I_{T(dc)}$	THD_i	PF
30^o						
60^o						
90^o						
120^o						

Also, generate a table to plot the control characteristic $I_{\alpha,eff} / I_{\alpha=0,eff}$ vs α.

DISCUSSION

You were presented with PSpice modeling and simulation of a single-phase AC voltage controller. A practical controller must have snubber circuits for di/dt and dv/dt protection.

Key point of the laboratory:

- An ac voltage controller is normally used in applications requiring limited voltage control, e.g., heater, light, etc.
- The input and the output voltage of AC voltage controller contain a high amount of harmonics, especially at the high delay angle.

Report:

1. Present all recorded waveforms and tables and discuss all significant points.
2. Discuss the effect of L on the load current, PF, and input current.

LAB 8: DESIGN OF AC VOLTAGE CONTROLLER - EXPERIMENT

LEARNING OBJECTIVES

After mastering this unit you will:

- Know the operation of an AC voltage controller and its performance parameters.
- Know the techniques of generating the control signals for ac voltage controllers.
- Be able to model and simulate ac voltage controllers.
- Be able to model and simulate control signals for ac voltage controllers.

READING ASSIGNMENT

Read sections 11.4 and 11.12 in chapter 11 of the textbook.

THEORY

R - LOAD

The circuit diagram of a single-phase ac voltage controller shown in Fig. 7-1. The performance parameters of an AC voltage controller are given by:

- Supply frequency, f_s.
- RMS input voltage, V_s.
- Load resistance, R.
- The RMS output (load) voltage is given by: $V_{o(rms)} = V_s \left[\frac{1}{\pi} \left(\pi - \alpha + \frac{\sin 2\alpha}{2} \right) \right]^{1/2}$.
- The RMS output (load) current, $I_{o,(rms)} = V_{o(rms)} / R$.
- Output power, $P_o = RI^2_{o(rms)}$.

RL - LOAD

- The current in T1 is given by: $i_1 = \frac{\sqrt{2}Vs}{Z}\left[\sin(\omega t - \theta) - \sin(\alpha - \theta) e^{(R/L)(\alpha/\omega - t)}\right]$. Where α is the delay (firing) angle and θ is the load angle $\theta = \tan^{-1}(\omega L / R)$.
- The extinction angle, β, can be determined from the transcendental equation: $\sin(\beta - \theta) = \sin(\alpha - \theta) \cdot e^{(R/L)(\alpha - \beta)/\omega}$.
- The conduction angle is defined as: δ = β-α.
- The RMS output voltage: $V_{o(rms)} = V_s \left[\frac{1}{\pi} \left(\beta - \alpha + \frac{\sin 2\alpha}{2} - \frac{\sin 2\beta}{2} \right) \right]^{1/2}$.
- The RMS thyristor current I_R: $I_R = \frac{V_s}{Z} \left[\frac{1}{\pi} \int_\alpha^\beta \left\{ \sin(\omega t - \theta) - \sin(\alpha - \theta) \cdot e^{(R/L)(\alpha/\omega - t)} \right\}^2 \cdot d(\omega t) \right]^{1/2}$.
- The RMS output current $I_{o(rms)}$: $I_{o(rms)} = \sqrt{2} I_R$
- The average value of thyristor current:

$$i_1 = \frac{\sqrt{2}Vs}{2\pi \cdot Z}\int_{\alpha}^{\beta}\left[\sin(\omega t-\theta)-\sin(\alpha-\theta)e^{(R/L)(\alpha/\omega - t)}\right]\cdot d(\omega t).$$

With AC power switch, a load can be connected to an AC system or disconnected from it. Such a power switch can be realized electrically, e.g. by two parallel thyristors connected in opposition – a so called pair of antiparallel arms (abbreviation W1). At small or medium powers a Triac may be used instead of the pair of antiparallel arms. Fig. 8-1 shows the single-phase AC voltage controller.

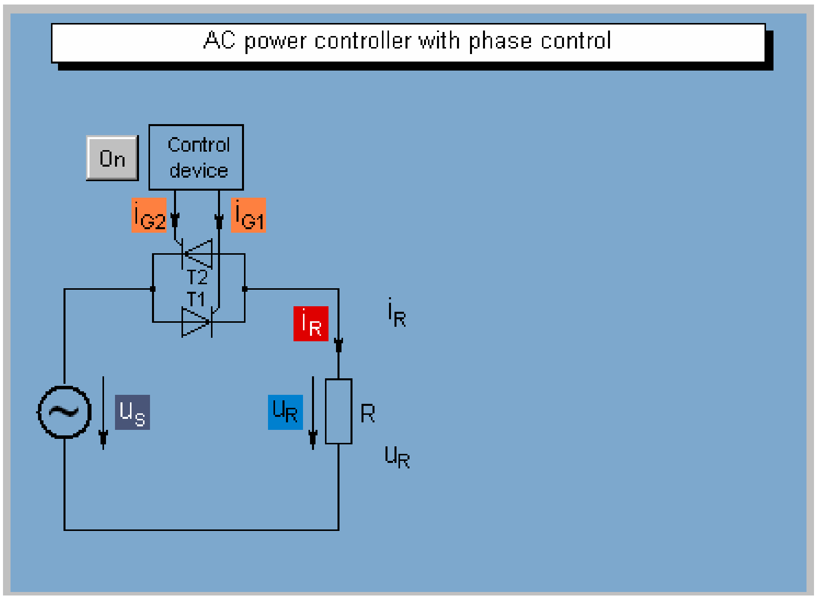


Figure 8-1 Single phase ac voltage controller

In the following experiment, the switching of alternating current via two thyristors connected as a pair of antiparallel arms will be studied. The load in this experiment is assumed to be purely ohmic (incandescent lamp). Fig. 8-2 shows AC voltage controller (ohmic load).

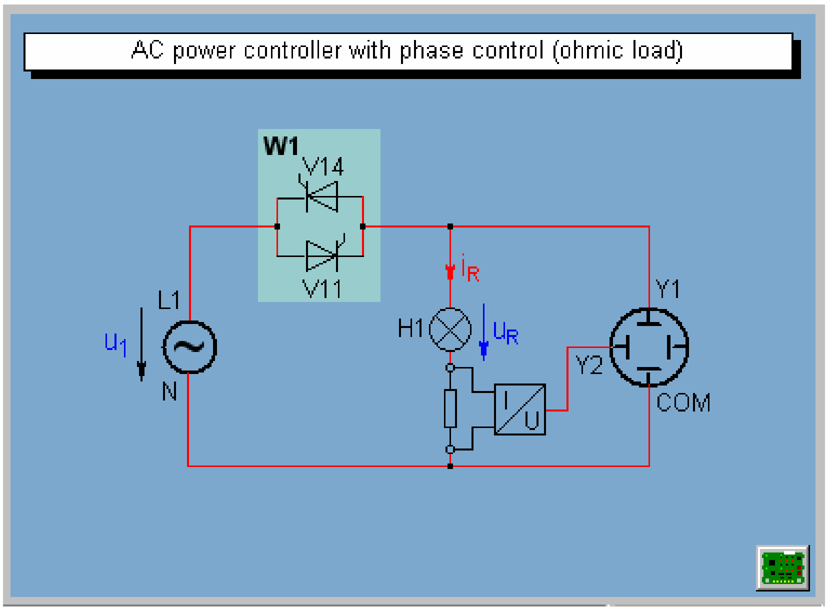


Figure 8-2 AC voltage controller – Ohmic load

The COM3LAB Board for AC voltage controller circuit high lighted by yellow boxes is shown in Fig. 8-3.

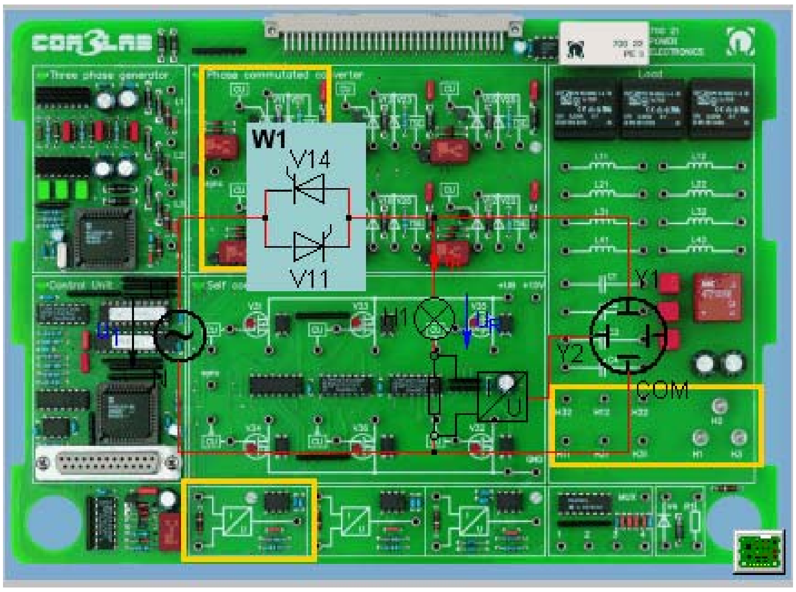


Figure 8-3 COM3LAB Board for AC voltage controller circuit

The larger trigger delay angle α, the later the thyristors are triggered, i.e., the smaller is the remaining part of the positive or the negative half-wave. The effective power converted at the load therefore decreases with increasing triggering delay angle. As the load in the experiment was ohmic, the shape of the load voltage and the load current are the same, i.e., they are in phase. Fig. 8-4 shows the load voltage and the load current waveforms for the delay angle α = 30^o.

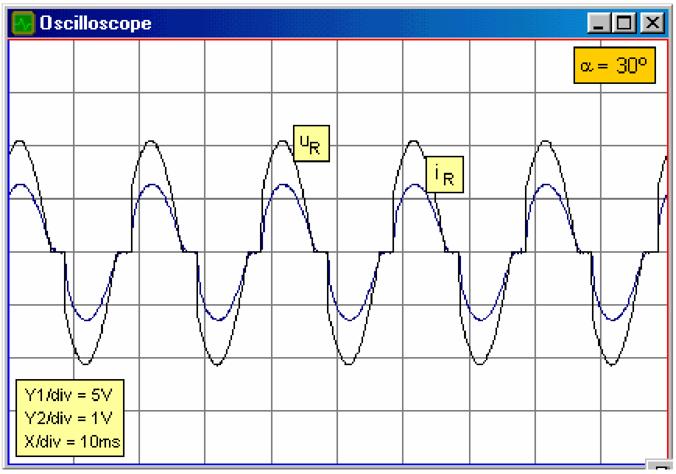


Figure 8-4 Load voltage and the load current waveforms for the delay angle α = 30^o

Frequency Spectrum:

In general, the resulting load voltages and currents are not sinusoidal when AC power controllers with phase control are used. In term of Fourier series introduced in course power electronics that means the frequency spectrum of the signals does not just contain the line frequency, but can also contain harmonics whose frequencies are integer multiples of the line frequency.

Open the Converter Analyzer with the button select the operating mode **disc**, and set the frequency range so that **fmax** = 600. Determine the spectrum for the trigger delay angle α = 90°. Fig. 8-5 shows the frequency spectrum for the delay angle α = 90°.

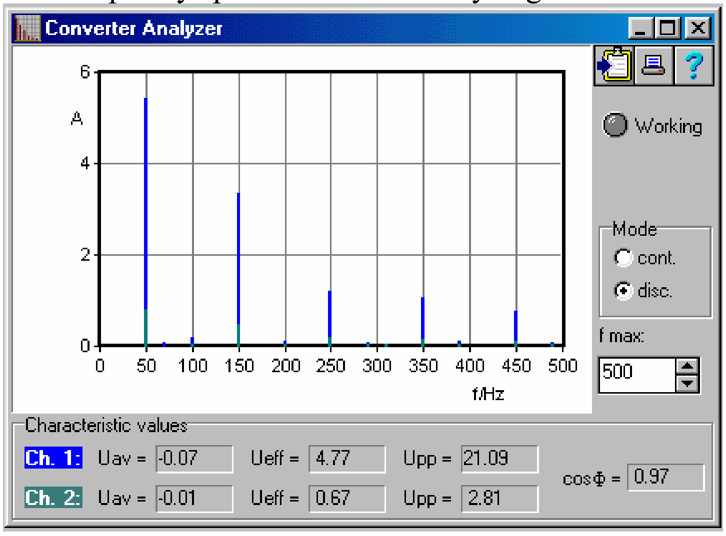


Figure 8-5 Frequency spectrum for the delay angle α = 90°

Control Characteristic

The control characteristic of the AC power controller with phase control and ohmic load will be recorded. Each time the RMS of the current recorded with the **Converter Analyzer**. Then the characteristic is displayed with the characteristic plotter. Remark: the RMS values can dragged from the **Converter Analyzer** to the **Characteristic Plotter** with Drag&Drop.
Now determine the RMS values of the load current (or of the measurement voltage, which is proportional to it) for triggering delay angle between 0° and 180° in step of 20° and drag the result into the **Characteristic Plotter**. Fig. 8-6 shows characteristic curve plotter. As result, the transmitted power decreases with increasing trigger delay angle α; and the control characteristic therefore drops continuously. The shape can be exactly calculated in the present case. The resulting formula reads $\frac{I_{\alpha,eff}}{I_{\alpha=0,eff}} = \sqrt{\frac{1}{\pi}\left(\pi - \alpha + \frac{1}{2}\sin 2\alpha\right)}$ and holds for trigger delay angles between 0° and 180°.

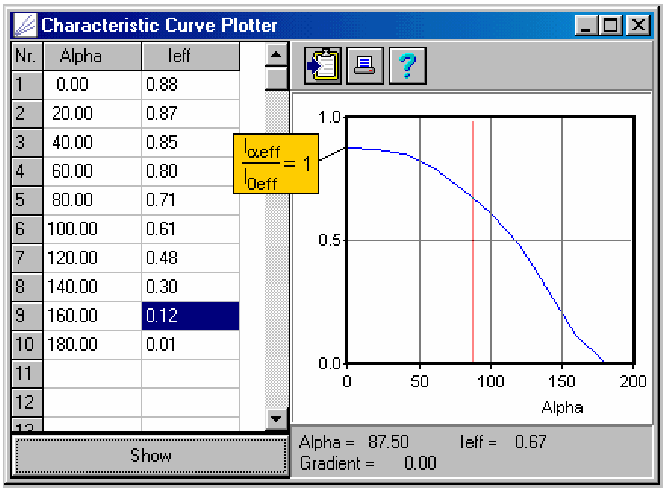


Figure 8-6 AC power controller with phase control and ohmic loal

Ohmic-Inductive Load

In the case of an ohmic-inductive load, the load current lags behind the load voltage by the load angle φ of about 60°. In the following circuit experiment the effect of the load angle on the controllability of the AC power controller will be studied. The load is again, a series connection of the incandescent lamp H1 and the transformer winding L11.
Setup the experiment as shown in Fig. 8-7. Open the control panel of the power converter control device and select the operating mode **PCC** (phase-commutated converter), operating mode **W1**.

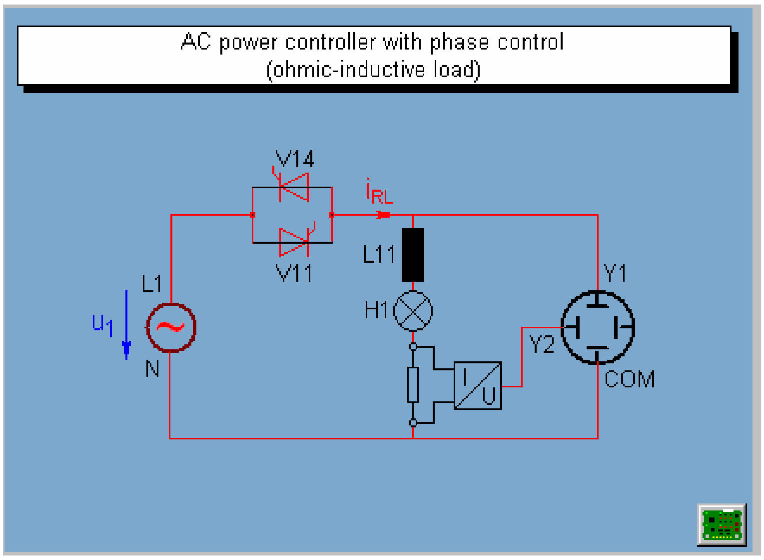
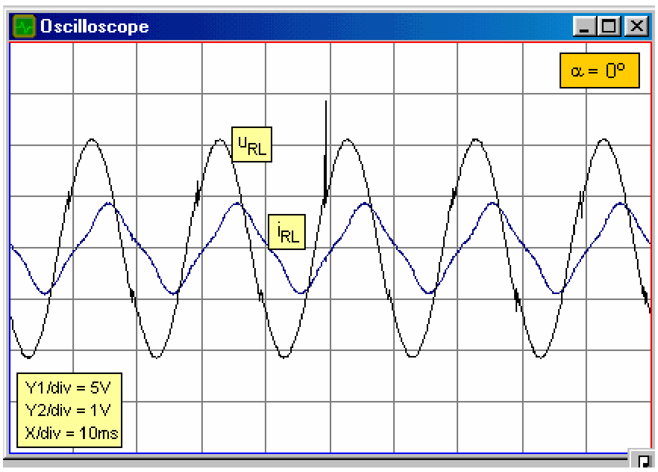


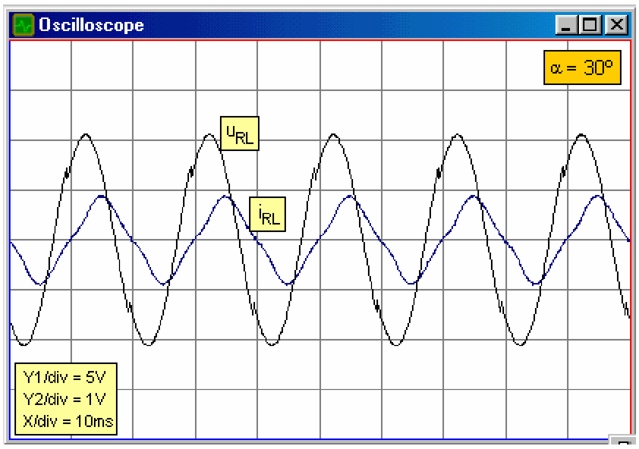
Fig. 8-7 AC power controller for ohmic-inductive load

Results

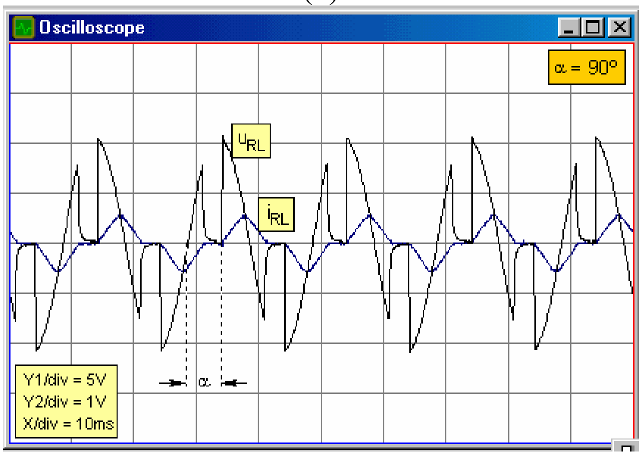
As long as the trigger control angle α is smaller than the load angle φ, there is unity control factor setting. This is the case for α = 0° and 30°. At a trigger control angle of 90°, there is reduced control factor setting. During fraction of the period no current flows and the load voltage suddenly drops to zero at the end of the conduction intervals. As a result, the blocking voltages at the thyristors rise with the same steepness. Fig. 8-8 (a, b, & c) shows load voltage and current waveforms for the delay angles mentioned above.



(a)



(b)



(c)

Figure 8-8 AC power controller-inductive load for three different angle α

Control Characteristic

Now determine the RMS values of the load current (or of the measurement voltage, which is proportional to it) for triggering delay angle between 0^o and 180^o in step of 20^o and drag the result into the **Characteristic Plotter**.

Result

Up to the load angle φ (approximately 60^o in this case) the control characteristic is constant; this is the range of unity control factor setting. If the trigger delay angle is further increased, the AC power controller operates with reduced control factor setting; the rms value of the current decreases continuously with increasing trigger delay angle. Fig. 8-9 illustrates the phenomena.

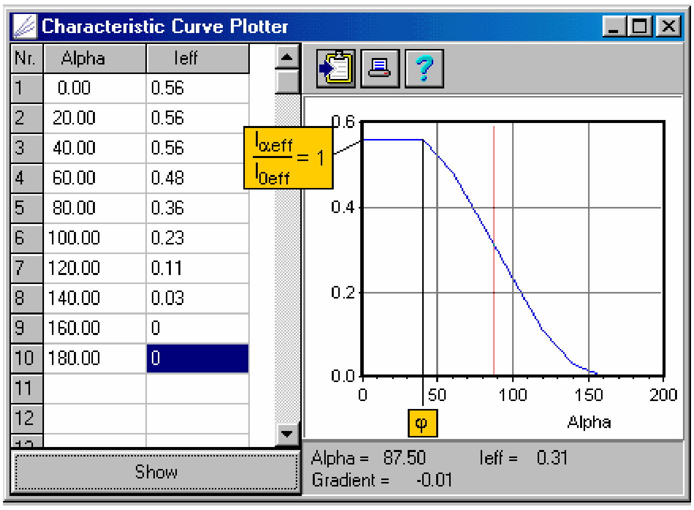


Figure 8-9 Control characteristic of AC voltage controller for inductive load

The rms output voltage will depend upon the delay angle or delay time (firing angle). Repeat simulation for the three values of delay angles and complete Table 8-1.

Table 8-1 Resistive Load

Delay angle α	$V_{o(rms)}$ RMS(V(4))	$I_{0(rms)}$ RMS(I(VX))	$I_{T(rms)}$	$I_{T(dc)}$	THD_i	PF
30°						
60°						
90°						
120°						

Repeat the PSpice simulation for RL-load (H11 & L11) and record all results in Table 8-2.

Table 8-2 Inductive Load

Delay angle α	$V_{o(rms)}$ RMS(V(4))	$I_{0(rms)}$ RMS(I(VX))	$I_{T(rms)}$	$I_{T(dc)}$	THD_i	PF
30°						
60°						
90°						
120°						

Also, generate a table to plot the control characteristic $I_{\alpha,eff} / I_{\alpha=0,eff}$ vs α.

Report:

1. Present all recorded waveforms and tables and discuss all significant points.
2. Compare your measurement results to your PSpice simulation in LAB7.
3. Discuss the effect of L on the load current, PF, and input current.

LAB 9: DESIGN OF A DC CHOPPER

LEARNING OBJECTIVES

After mastering this unit you will:

- Know the operation of a DC chopper and its performance parameters.
- Know the guidelines for designing an output LC-filter for DC choppers.
- Know how to model and simulate DC chopper circuits.
- Be able to model a DC pulse source.

READING ASSIGNMENT

'.

Read sections 5.3 and 5.12 in chapter 5 of the textbook.

THEORY

The circuit diagram of a step down chopper is shown in Fig. 9-1. The performance parameters of the chopper are given by:

- Chopping frequency, f_s.
- Chopping period, T_s.
- On time of the chopper, t_{on}.
- Duty cycle, $k = t_{on}/T_s$.
- DC input voltage, V_s.
- Average output (load) voltage, $V_{o(dc)} = kV_s$.
- Average output (load) current, $I_{o(dc)} = V_{o(dc)}/R = kV_s/R$.
- DC input current, $I_{s(dc)} = kI_{o(dc)}$.
- Output DC power, $P_{o(dc)} = V_{o(dc)} . I_{o(dc)}$.

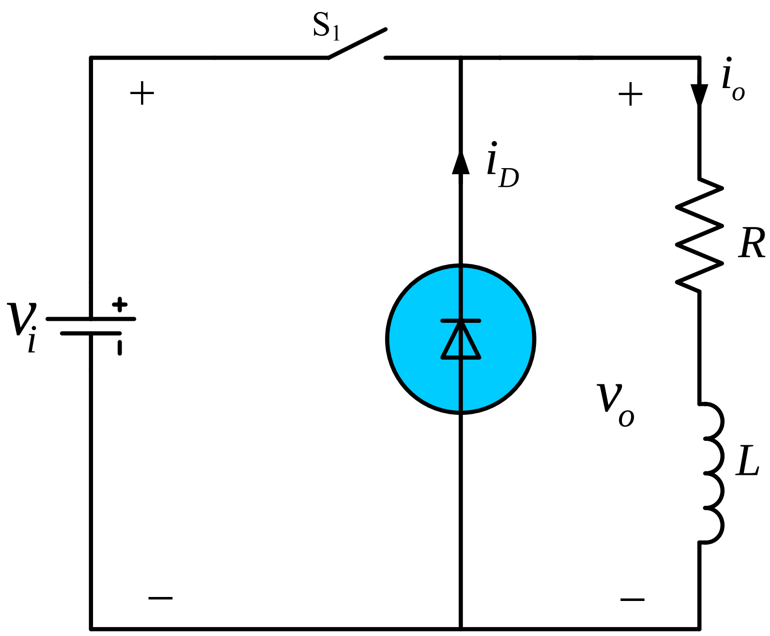


Figure 9-1 DC chopper with RL-Load

Design Specifications

f_s = 20 kHz
$T_s = l/f_s$ = 50 μs, and $t_{on} = kT_s$.
$V_{o(dc)} = kV_s$
which, for V_s = 15 V and $V_{o(dc)}$ = 7.5 V, gives the required value of k = 0.5 or 50%.
Average output (load) current, $I_{o(dc)} = V_{o(dc)}/R$.
DC input current, $I_{s(dc)} = k\, I_{o(dc)}$
Output DC power, $P_{o(dc)} = V_{o(dc)}\, I_{o(dc)}$.

Transistor:

Average transistor current, $I_{Q(dC)} = kI_{o(dc)}$.
RMS transistor current, $I_{Q(rms)} = \sqrt{k}\, I_{o(dc)}$.
Peak transistor current, $I_{Q(pcak)}$ =
Peak transistor voltage, $V_{Q(peak)} = V_s$ = 15 V. Choose 50 V.

Diode:

Average diode current, $I_{D(dc)} = k\, I_{o(dc)}$.
RMS diode current, $I_{D(rms)} = \sqrt{k}\, I_{o(dc)}$.
Peak diode current, $I_{D(peak)}$.
Peak diode voltage, $V_{D(peak)} = V_s$ = 15V. Choose 50 V.

PSpice SIMULATION

```
LAB 9 DC Chopper Circuit
VS   1   0   DC   15V
Vg   6   0   PULSE (0V   10V   0  1NS  1NS  0.5MS   1MS)
Rg   6   0   10MEG
R    3   4   17
L    4   5   22.3MH
VX   5   0   DC   0V   ; Load battery voltage
VY   2   3   DC   0V   ; Voltage source to measure chopper current
DM   0   3   DMOD              ; Free-wheeling diode
.MODEL  DMOD  D(IS=2.22E-15 BV=1200V CJO=1PF TT=0) ; diode model
S1   1   2   6   0   SMOD       ; Switch
.MODEL  SMOD  VSWITCH (RON=0.01 ROFF=10E+6 VON=10V VOFF=5V)
.TRAN   10US   10MS   8MS          ; Transient analysis
.PROBE                          ; Graphics post-processor
.OPTIONS ABSTOL = 1.00N  RELTOL = 0.01  VNTOL = 0.1  ITL5=40000
.FOUR   1KHZ   I(VX)   I(VY)        ; Fourier analysis
.END
```

RESULTS

1. Run the simulation of Fig 9-1 and complete the Table 9-1 for three values of load resistances and L42 = 22.3mH for k = 0.5.

Table 9-1

R (Ω)	$V_{o(dc)}$	$I_{0(dc)}$	$I_{L(dc)}$	ΔI_L	$P_{o(dc)}$	V_sI_s
15 (H11)						
30 (H11+H22)						
45 (H11+H22+H33)						

2. A DC chopper can be used to obtain a variable output voltage which can control the speed of a DC motor or the variable DC link voltage to an inverter. The PSpice plots of the load current and the load voltage are shown in Fig. 9-2.
3.

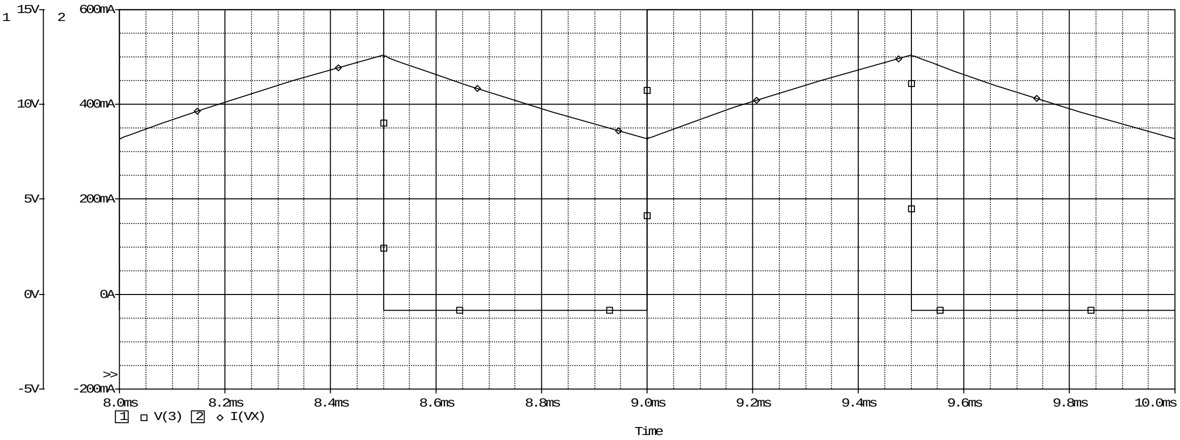


Figure 9-2 Load current and the load voltage of a DC chopper

Run the simulation of Fig 9-1 and complete the Table 9-2 for different values of duty cycle. Comment on the effects of duty-cycle on the load current ripple ΔI_L. Use R(H11) and L(L42).

Table 9-2

k	$V_{o(dc)}$ (V)	$I_{0(dc)}$ (mA)	ΔI_L (mA)	$I_{L(dc)}$ (mA)	$P_{o(dc)}$ (W)	$I_{s(dc)}$ (mA)	V_sI_s (W)
0.0	0	0	0	0	0	0	0
0.1							
0.2							
0.3							
0.4							
0.5							
0.6							
0.7							
0.8							
0.9							
1.0	15		0				

DISCUSSION

You were presented with the PSPICE modeling and simulation of a DC chopper. A DC chopper can serve as a variable DC source to a DC load or as the variable DC link voltage to an inverter.

Key points of the laboratory:

- A DC chopper can be used for stepping up or stepping down a DC voltage. It may be regarded as a DC transformer.
- A DC chopper can be used as a voltage regulator *in* order to maintain a regulated DC output voltage. An output *filter is* used to filter out the ripple contents of the output voltage and current. The chopper is generally operated under closed-loop control to maintain the output voltage at a desired value.
- A DC chopper can also be used to give a variable DC output voltage for the speed control of Dc motors or the variable DC link voltage to an inverter. The chopper is generally operated under closed-loop control to maintain the control variable' at a desired value. The output variable could be the speed (or position) of the motor or the output DC link voltage.
- The output voltage contains harmonics. An output filter is normally connected to obtain a smoother DC output voltage.
- The input current is a pulsating type and contains harmonics. An input filter is normally connected to smooth the input current and to provide a low impedance path to the DC source.
- The DC chopper should be operated at as high a frequency as possible because it will reduce the sizes of filter components.

EXPERIMRNT

DC Choppers - Buck Converter

The buck converter enables a direct voltage to be lowered. This is achieved by switching the input voltage U_o on and off at a high frequency with the aid of a DC power switch (GS), which in practice can be, e.g., a MOSFET. As a result there is a direct voltage U_{RL} at the load whose mean value U_{AV} depends on the ratio of the operating intervals and the break times. Fig. 9-3 shows the buck converter with PWM "pulse width modulation".

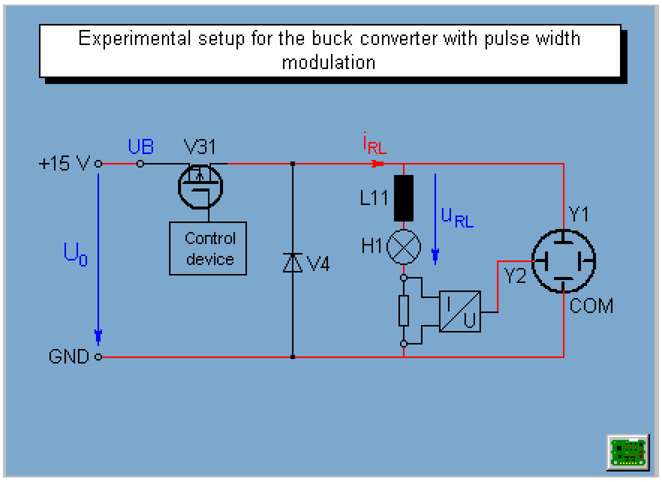


Figure 9-3 Buck converter with pulse width modulation

The circuit elements and devices of the COM3LAB board are shown in Fig. 9-4.

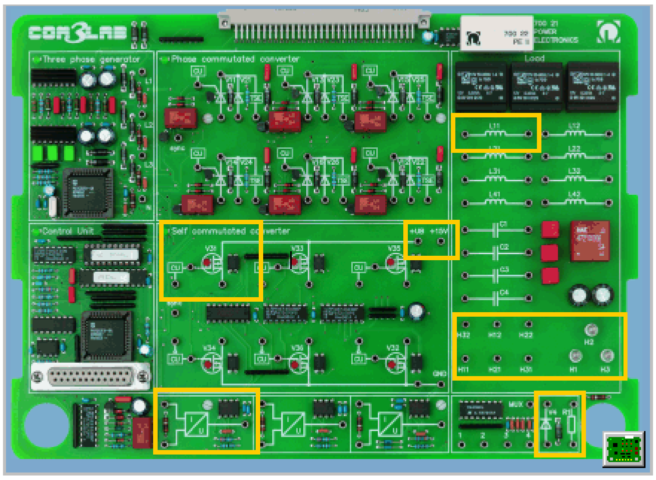
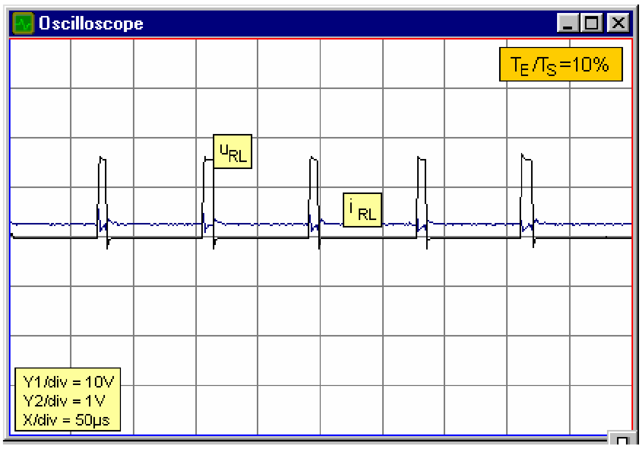


Figure 9-4 Elements and devices marked by yellow boxes for buck dc chopper

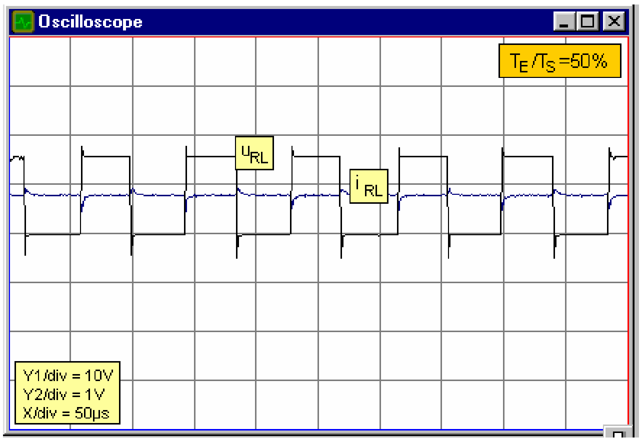
In the following experiment the principle of the buck converter with pulse width control will be studied. The output voltage is controlled at a constant clock-pulse rate (that is at a constant period Ts) by varying the width of the switch on pulses (operating interval T_E). The control pulses for the MOSFET, which functions as a DC power switch, are generated automatically by the COM3AB control device.

Result:

As the operating voltage U_B, i.e. the input voltage of the converter U_o, is 15V, a pulse control factor of 10% leads to a mean value of the output voltage of about 1.5V, whereas a pulse control factor of 50% leads to a voltage of about 7.5V. The mean value of the load current, too, is proportional to the pulse control factor. The results of voltage and current output buck DC chopper are shown in Fig. 9-5 (a , b)



(a)



(b)

Figure 9-5 Voltage & current output wave forms of buck DC chopper

Pulse Frequency Control

Instead of varying the operating interval T_E at a constant period Ts as in the case of pulse width control, you can also vary Ts at a constant operating time. In this procedure, which is called pulse frequency control, the mean value U_{AV} is also proportional to the ratio of T_E and Ts. However, pulse frequency control is applied very seldom in practice. Fig. 6-9 shows the buck converter with pulse frequency control.

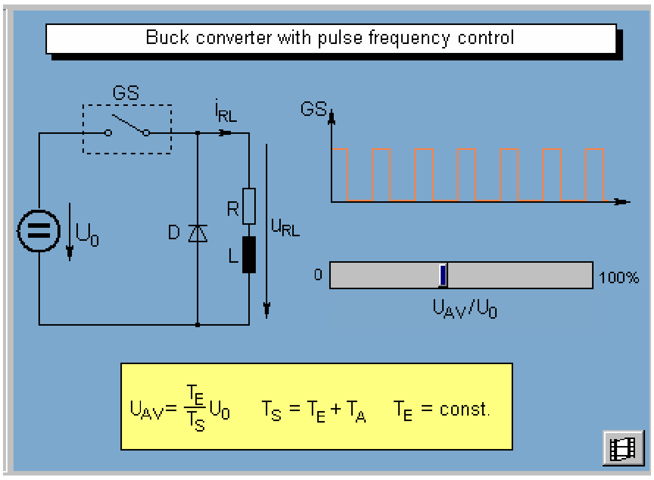


Figure 9-6 Buck converter with pulse frequency control

Construct the circuit in Fig. 6-3 and tabulate the results based on Table 9-1 and 9-2.

<u>Report:</u>

1. Present all recorded waveforms and discuss all significant points.
2. Compare the PSpice and experiment results, write your comments?

LAB 10: THREE-PHASE VOLTAGE SOURCE INVERTER

LEARNING OBJECTIVES

After mastering this unit you will:

- Know the operation of a three-phase PWM inverter.
- Know the techniques of generating the control signals for PWM inverters.
- Be able to model and simulate PWM inverter circuits.
- Be able to verify the simulation by running the experiment.

READING ASSIGNMENT

Read sections 6-5 in chapter 6 of the textbook.

THEORY

The circuit diagram of a single-phase voltage source inverter is shown in Fig. 10-1. Two types of control signals can be applied to transistors: 180^o conduction or 120^o conduction. The 180^o conduction has better utilization of the switches and is the preferred method. In 180 degree conduction, each transistor conducts for 180 degree. Three transistors remain on at any instant of time. When transistor Q1 is switched on, terminal **a** is connected to the positive terminal of the dc input voltage. When transistor Q4 is switched on, terminal **a** is brought to the negative terminal of the dc source. There are six modes of operation in a cycle and the duration of each mode is 60^o. The transistors are numbered in the sequence of gating the transistors (e.g., 123, 234, 345, 456, 561, 612).

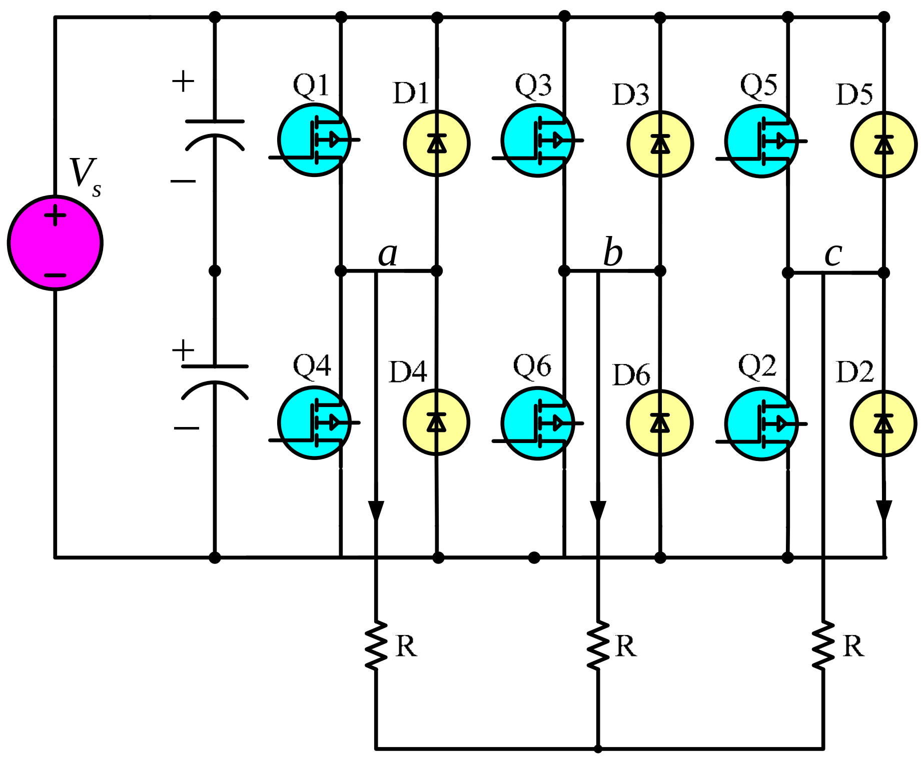


Figure 10-1 Three-phase voltage source inverter

Y- Connected R-Load

The instantaneous line to line voltage v_{ab} can be expressed in A Fourier series,

$$v_{ab} = \frac{a_0}{2} + \sum_{n=1}^{\infty} (a_n \cdot \cos(n\omega t) + b_n \cdot \sin(n\omega t))$$

Due to the quarter-wave symmetry along the x-axis, both a_0 and a_n are zero. Assuming symmetry along the y-axis at ωt = π/6, we can write b_n,

$$b_n = \frac{1}{\pi}\left[\int_{-5\pi/6}^{-\pi/6} -V_s \cdot d(\omega t) + \int_{\pi/6}^{5\pi/6} V_s \cdot d(\omega t)\right] = \frac{4V_s}{n\pi}\sin\left(\frac{n\pi}{2}\right)\cdot\sin\left(\frac{n\pi}{3}\right)$$

Which, reorganizing that v_{ab} is phase-shifted by π/6 and the even harmonics are zero, gives the instantaneous line-to-line voltage v_{ab} as

$$v_{ab} = \sum_{n=1,3,5,\ldots}^{\infty} \frac{4V_s}{n\pi}\cdot\sin\left(\frac{n\pi}{3}\right)\cdot\sin n\left(\omega t + \frac{\pi}{6}\right)$$

Both v_{bc} and v_{ca} can be found by phase shifting v_{ab} by 120° and 240°, respectively.

$$v_{ab} = \sum_{n=1,3,5,\ldots}^{\infty} \frac{4V_s}{n\pi}\cdot\sin\left(\frac{n\pi}{3}\right)\cdot\sin n\left(\omega t - \frac{\pi}{2}\right)$$

$$v_{ab} = \sum_{n=1,3,5,\ldots}^{\infty} \frac{4V_s}{n\pi}\cdot\sin\left(\frac{n\pi}{3}\right)\cdot\sin n\left(\omega t - \frac{7\pi}{6}\right)$$

The line-to-line rms voltage can be found from:

$$V_L = \left[\frac{2}{2\pi}\int_0^{2\pi/3} V_s^2 \cdot d(\omega t)\right]^{1/2} = \sqrt{\frac{2}{3}}V_s = 0.8165 \cdot V_s$$

The rms nth component of the line voltage is:

$$V_{Ln} = \frac{4V_s}{\sqrt{2}\cdot n\pi}\cdot\sin\left(\frac{n\pi}{3}\right)$$

Which, for n = 1, gives the rms fundamental line voltage:

$$V_{L1} = \frac{4V_s}{\sqrt{2}\cdot\pi}\cdot\sin(60) = 0.7797 \cdot V_s$$

The rms value of line-to-neutral voltages can be found from the line voltage,

$$V_p = \frac{V_L}{3} = \frac{\sqrt{2}V_s}{3} = 0.4714 \cdot V_s$$

PSpice SIMULATION

Type the PSpice program below and then save it as lab10.cir file.

```
Lab10: Three-Phase Inverter
VS   1   0   DC   15V
RB1  22   6   50
Rg1  22   3   10MEG                ; Control voltage
Vg1  22   3   PULSE (0  10V  0  1NS  1NS  0.5MS  1MS)
Rb2  16  15   50
Rg2  16   0   10MEG
Vg2  16   0   PULSE (0  10V  166.67US  1NS  1NS  0.5MS  1MS)
Rb3   8   7   50
```

```
Rg3   8    4    10MEG
Vg3   8    4    PULSE (0  10V  333.33US  1NS  1NS  0.5MS  1MS)
Rb4  12   11    50
Rg4  12    0    10MEG
Vg4  12    0    PULSE (0  10V  500US  1NS  1NS  0.5MS  1MS)
Rb5  10    9    50
Rg5  10    5    10MEG
Vg5  10    5    PULSE (0  10V  666.67US  1NS  1NS  0.5MS  1MS)
Rb6  14   13    50
Rg6  14    0    10MEG
Vg6  14    0    PULSE (0  10V  833.33US  1NS  1NS  0.5MS  1MS)
VY    1    2    DC    0V    ; Voltage source to measure supply current
VX    3   20    DC    0V    ; Measures load phase current
RA   20   21    15
*LA   17   21    5MH        ; LA is included
RB    4   21    15
*LB   18   21    5MH        ; LB is included
RC    5   21    15
*LC   19   21    5MH        ; LC is included
D1    3    2    DMOD        ; Diode
D3    4    2    DMOD        ; Diode
D5    5    2    DMOD        ; Diode
D2    0    5    DMOD        ; Diode
D4    0    3    DMOD        ; Diode
D6    0    4    DMOD        ; Diode
.MODEL DMOD D(IS=2.22E-15 BV=1200V IBV=13E-3 CJO=0 TT=0) ; diode model
Q1    2    6    3    3    2N6546     ; BJT Switch
Q3    2    7    4    4    2N6546     ; BJT Switch
Q5    2    9    5    5    2N6546     ; BJT Switch
Q2    5   15    0    0    2N6546     ; BJT Switch
Q4    3   11    0    0    2N6546     ; BJT Switch
Q6    4   13    0    0    2N6546     ; BJT Switch

.MODEL 2N6546 NPN(IS=2.33E-27 BF=13 CJE=1PF CJC=607.3PF TF=26.5NS)
*.MODEL  2N6546  NPN (Is=6.734f Bf=416.4 Ise=6.734f Br=.7371
*+    Cjc=3.638p Mjc=.3085 Vjc=.75 Cje=4.493p Mje=.2593 Vje=.75
*+    Tr=239.5n Tf=301.2p)           ; BJT parameters
.TRAN    5US   2.5MS  1.0MS          ; Transient analysis
.PROBE                               ; Graphics post-processor
.options abstol = 1.00n  reltol = 0.01  vntol = 0.1  ITL5=20000
.FOUR   1KHZ   I(VX)   V(3,21)       ; Fourier analysis
.END
```

1. If you are in the Program Manager of the Window, double click the left mouse button on the Design Center icon to open the Design Center Group (showing Schematics, PSpice, Probe, Parts).
2. Double click the left mouse on the PSpice icon to open the PSpice menu (showing File, Display, Help).
3. Choose Open from the File menu and type the file name, say lab10.CIR. Do not forget to specify the drive in which the file resides, e.g. d:\EE460\lab10.cir. PSpice will automatically run the simulation.

4. Choose Run Probe from the File menu.
5. Choose Trace from the Probe menu.
6. Select Add from the Probe menu and plot by choosing, the plot variable, the output current, -I(R).

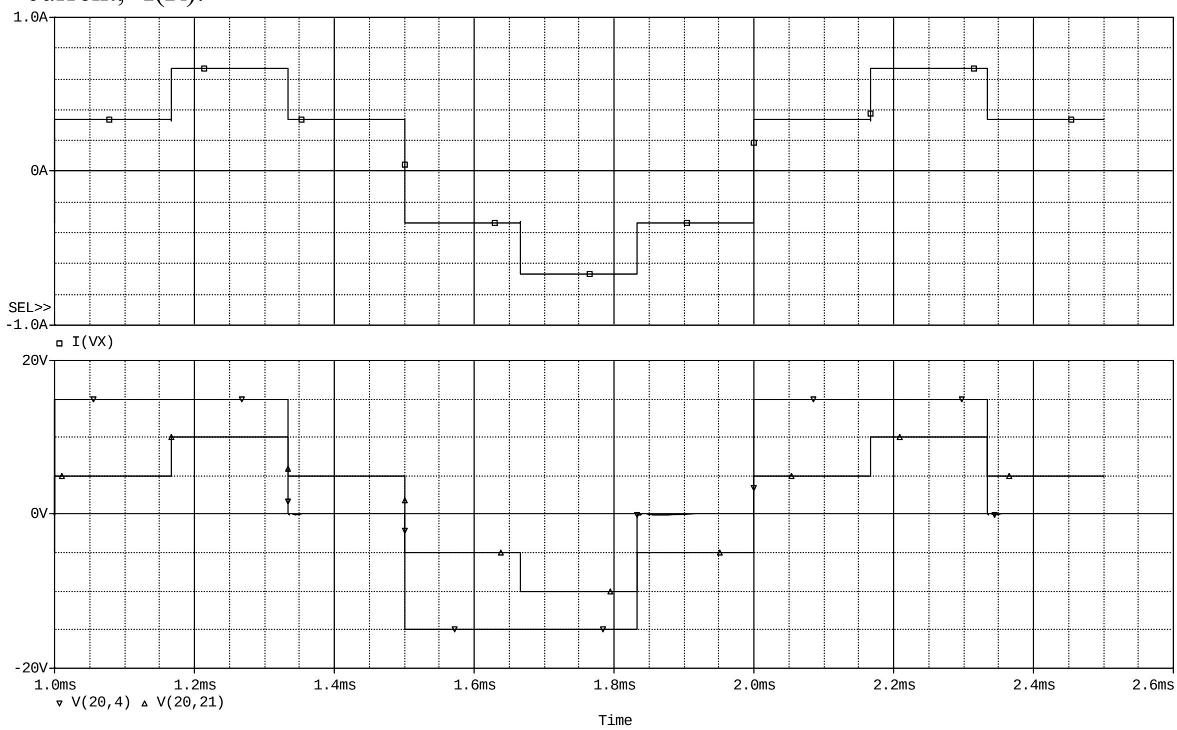


Figure 10-2 Output voltage waveform and control signals for R – load

RESULTS

1. Run the simulation of Fig 10-1 and complete the Table 10-1 for resistive load.

Table 10-1

R (H11..H31)	$V_{o(rms)}$	$I_{o(rms)}$	V_{o1}	I_{o1}	$P_{out}=RI^2_{o(rms)}$	THD_i
15						

2. Run the simulation of Fig 10-1 and complete the Table 10-2 for inductive load (use L11, L21, and L31)

Table 10-2

R (H11..H31)	$V_{o(rms)}$	$I_{o(rms)}$	V_{o1}	I_{o1}	$P_{out}=RI^2_{o(rms)}$	THD_i
15						

3. Compare the two tables and comment on the results.

DISCUSSION

You were presented with the PSPICE modeling and simulation of a three-phase voltage source inverter. Also, the guidelines for determining the ratings of devices and components

are presented. An inverter can serve as a fixed (or variable) AC source or as an intermediate stage of a DC supply. A practical inverter must have snubber circuits for di/dt and dv/dt protection.

EXPERIMENT

Three-Phase Voltage Source Inverter

For supplying three-phase loads three-phase circuits are required. The smoothing capacitor in the link circuit of the converter is charged via the supply converter so that its voltage is approximately constant. Via the valves V1 to V6, the load-side converter switches this link voltage U_o cyclically to the stator-circuit terminals of the connected load, which in the case under consideration is balanced and ohmic-inductive. The transistors are triggered with a phase shift of 60^o one after another as shown in Fig. 10-3.

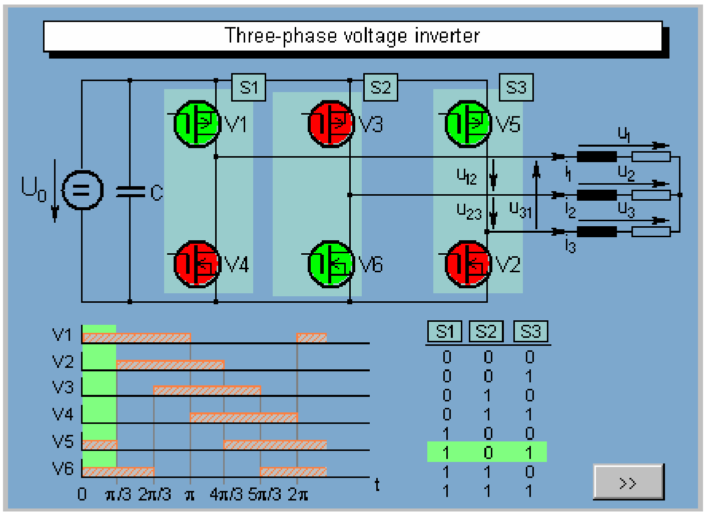


Figure 10-3 Three-phase voltage inverter

In the following experiment, the circuit states of the three-phase voltage will be closely studied. The individual circuit states are selected by hand via COM3LAB control device. Then phase-to-phase voltage u_{12} and the phase voltage u_1 are measured with the multimeters in each case.

Setup the experiment as shown in Fig. 10-4, open the control panel of the power converter control device, and select the operating mode **Phase**, commutation model **Block**.

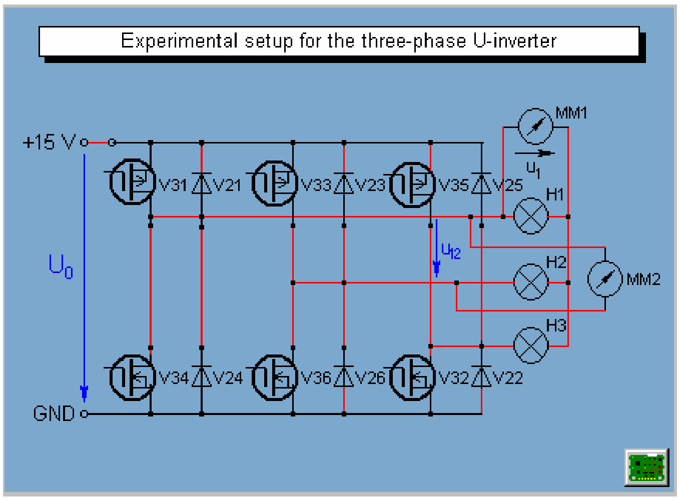


Figure 10-4 Three-phase voltage inverter with resistive load

Result

The phase-to-phase voltages alternate between U_o, 0 and U_o each, depending on which transistors are conductive in that moment. The shape of the phase voltages, too, is a direct result of the switching state of the transistors. Here the four voltage levels $\pm U_o/3$ and $\pm 2U_o/3$ have been assumed. There is a phase shift of 120^o or 240^o, respectively, between the individual phase-to-phase and phase voltages. Fig. 10-5 shows the phase control states and phase and phase-to-phase voltages.

Phase and phases-to-phase voltages in the case of the three-phase

S_1	S_2	S_3	u_1	u_{12}
0	0	1	$-U_0/3$	0
0	1	1	$-2U_0/3$	$-U_0$
0	1	0	$-U_0/3$	$-U_0$
1	1	0	$+U_0/3$	0
1	0	0	$+2U_0/3$	$+U_0$
1	0	1	$+U_0/3$	$+U_0$

Figure 10-5 Phase control states and phase and phase-to-phase voltages

Control Methods of the Inverter

In the following experiment, it will be investigated which of the methods described so far is at work in the COM3LAB inverter.

Result

The inverter operates as pulse-controlled inverter. At amplitude of 100%, there is unity control factor setting; if the amplitude is reduced, this leads to a reduction of the operating interval T_E and to an enhancement of the turn-off time T_A in accordance with the previous

definition of the pulse control factor $\lambda = \frac{T_E}{T_E + T_A}$, if the amplitude is set to 50%, both intervals are equal. Figs. 10-6 and 10-7 show the voltage output for different control factors.

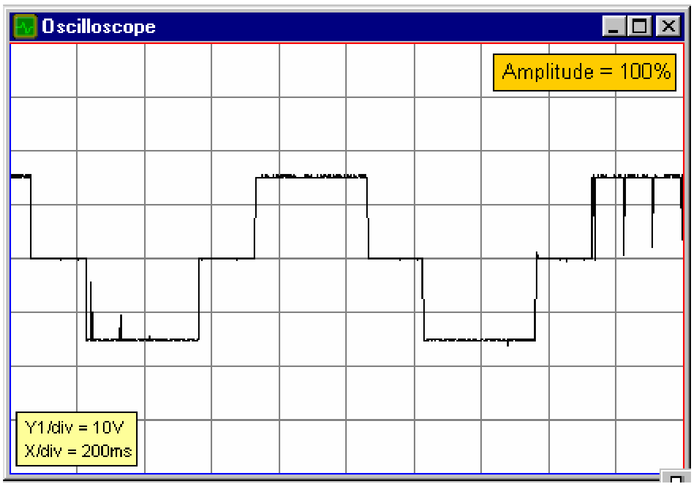


Figure 10-6 Pulse control of three-phase inverter for a unity control factor

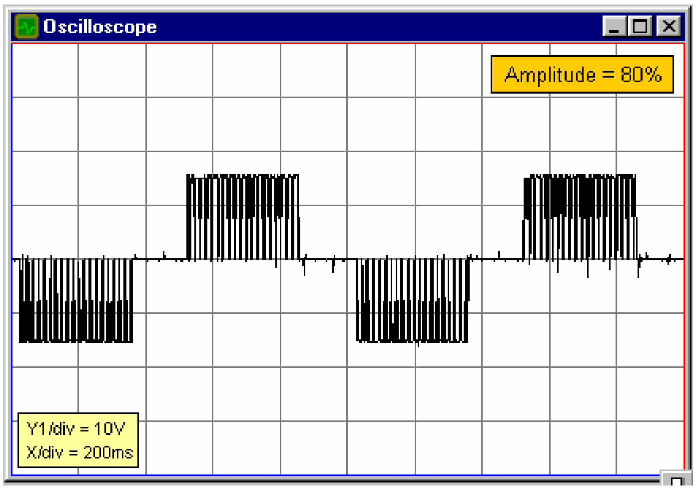


Figure 10-7 Pulse control of three-phase inverter for 80% control factor

Report:

1. Present all recorded waveforms and discuss all significant points.
2. Compare the PSpice and experiment results, write your comments?

APPENDIX

LAB 0: PSPICE TUTORIAL

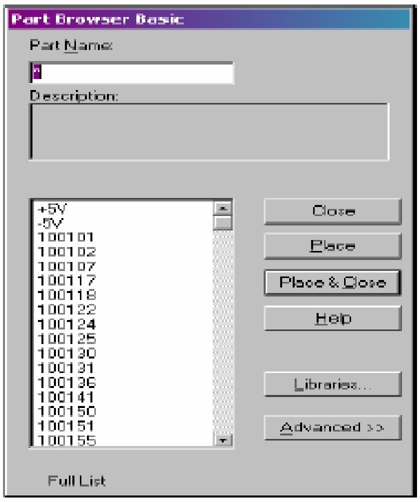
This tutorial is designed to show you how to use the PSpice circuit simulation form MicroSim with the schematic capture front end, Schematics. It will help you solve some of the problems you are given in class.

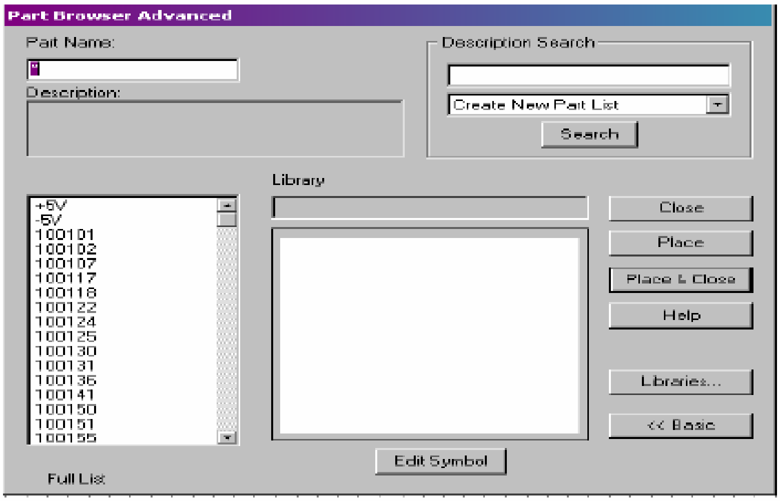
STARTING THE PROGRAM:

(1) From the **Start** menu, point to the **Micro Slim program**, and then select **Schematics**.
(2) We should see a schematics window; zoom in or out using the zoom icon according to your preference. Next step we are going to place parts.

PLACING PARTS:

Remember all parts in PSpice are retrieved using this procedure.
(1) Using the LEFT mouse button, LEFT click on the **Draw** selection in the schematics window. The draw menu will pull-down.
(2) LEFT click on **Get New Part** or hold and type **CTRL-G**. A part browser dialogue will appear that shows all the parts available to us. If you know the name of the part, type it in. For example, most voltage sources start with the letter v. If you type in the letter v, the list of parts will scroll down to display parts that begin with the letter v.





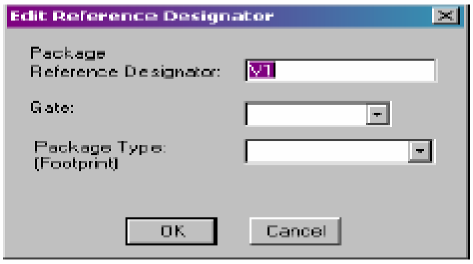
(3) To select a part and see it's description, LEFT click on the name of the part. To see what a part looks like, LEFT click on the **Advanced** button in the part browser. You should see an electrical representation drawing of the component.
(4) We wish to select a generic DC voltage supply. LEFT click or type in vdc in the browser. To place the part, LEFT click on **Place & Close** and the part with be attached to the mouse pointer. Note that the graphics in the program move with the mouse. Now with the part attached, LEFT click to place the part on the sheet.
Note again that automatically a second source appears on the mouse pointer, this is the auto repeat function. To **add** the same part, LEFT click on the mouse again.
To **stop** adding the same part, RIGHT click on the mouse. To rotate the part type and hold **CTRL-R**; rotate your part to any direction you want.

EDITING PARTS:

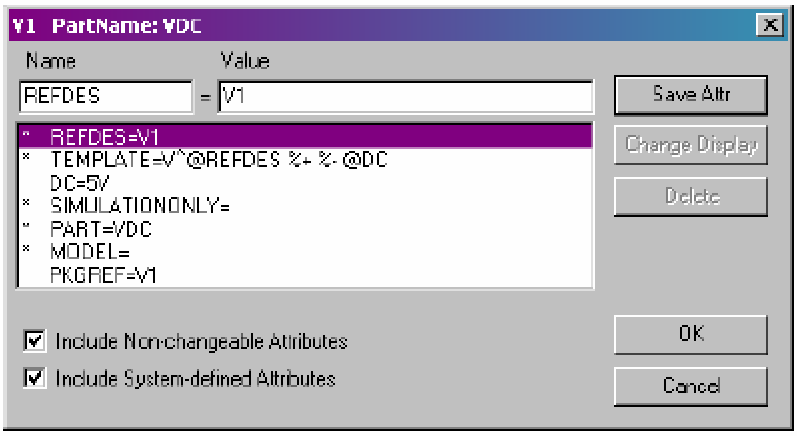
All parts and components have to be edited to fit the right attributes.

(1) Let's assume we want a 12 volts DC source on the part we placed from the above step.

a. First way is to do this, is to edit individual part attributes that are displayed on the sheet. To do this, double click the LEFT mouse button on the **DC =** or the **0V** (used by older Micro Slim versions) text next to the DC source on the schematic. A dialogue bow will appear saying we are **changing** or **setting** the DC attribute. Set the voltage to 12V and click on **OK**. To edit the name of the voltage source, double click the LEFT mouse button on the text V1 (the default voltage name). A dialog box called **Edit Reference Designator** will appear. Type in the desired name and click **OK**.



b. Second way to edit part attributes is by clicking on the source graphics. The graphics will be highlighted in red when selected properly. When the part is highlighted, click the LEFT mouse button on the **Edit menu selection**. Then click the LEFT button one more time on the **Attributes menu selection**. A dialog box appears, which can be used to change many of the attributes of any selected part. To change the voltage or the name of the DC source, we can LEFT click on the line **12V**; a text **DC** will appear in the **Name** box and the number **12** will appear in the **Value** box. For any of the changes to take effect, RIGHT click on the mouse on the **Save Attr button**.



WIRING COMPONENTS:

(1) Click on the LEFT mouse button on the **Draw menu selection**, and then LEFT click on the **Wire menu selection** or by using **CTRL-W** or by clicking on the **Draw wires icon**. A pencil appears in replacement of the pointer on the mouse. To start drawing a wire, LEFT click at the desired point and then move the pencil away. Note that the wire is dashed; this indicates that it is not yet a wire. If you missed the positive

terminal, click the RIGHT mouse button and start over. The dashed lines change to solid lines, indicating that the lines are wires.

(2) Click the LEFT button to make a connection and then move the mouse a way. To stop drawings wires, click the RIGHT button.

(3) Note that some connections have a black dot. This indicates a connection, it is not necessary to have a dot present when a wire joins a pin, but having the dot emphasizes that a connection is present. To make a dot appear at a pin, make the wire overlap the pin when you are drawing wires. If you draw to the tip of a pin, a dot will not appear. Dots are always drawn when a wire meets at a T-connection.

USING A PROBE:

(1) You should use a probe to plot traces. If you want to plot the voltage at a node or the current at a branch, you have to label that branch or node first. Run probe from the **Analysis menu** and then select **Add trace** from the menu bar (for newer PSpice versions) or the icon marked with a **V** and labeled **Voltage/Lever marker**. The left plane will list all the types of traces we can plot and the right plane displays all the mathematical operations we can perform on the traces. It lists digital waveforms, voltages, currents and alias names, which refer to different names of the same thing. In this course we usually select analog voltages, so in this case we select **Analog** from the list. You can add as many traces as possible to assist in a plot. If you want to trace a current you have to select **Currents** instead of Analog. You can remove individual traces by LEFT clicking on the name you want to delete and press the **Delete** key on your keyboard.

(2) **Ground your circuit**. To run any circuit on Pspice, you must have at least one ground in your circuit. To ground your circuit, you may place a part called **AGND** in you circuit, or type in "ground" in your **Get new part** browser and find the appropriate ground you need.

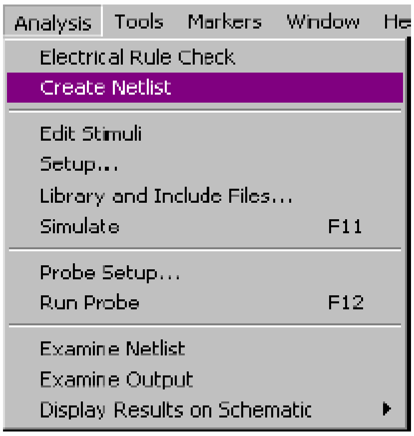
DC NODAL ANALYSIS:

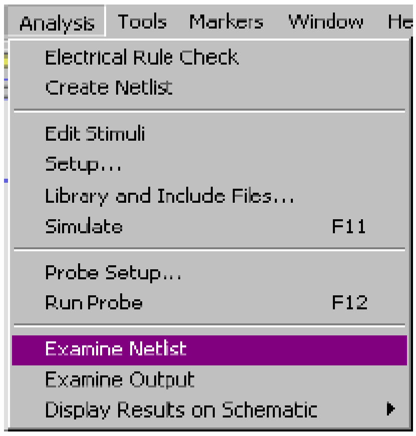
(1) The node voltage analysis performed by Pspice is for DC node voltages only. This analysis solves for the DC voltage at each node of the circuit. If any AC or transient sources are present in the circuit, those sources are set to zero. If you need to find AC node voltages, you can run an **AC sweep** from the **Analysis menu bar**.

(2) In order to do nodal analysis you also have to add two more parts into your circuit. **IPROBE** and **VIEWPOINT**. **IPROBE** can be placed on any branch in the circuit to view the current, after you do a simulation of the circuit. If you place **VIEWPOINT** at a node after simulation, you can view the voltage of the circuit at that node. Both parts can be found in the **Get new part** selection browser.

(3) After placing your parts, wiring them together and placing the appropriate nodal analysis probe desired we could now **Examine** the circuit to check for errors.

a. LEFT mouse click on the **Analysis menu bar**. LEFT click again on **Create Netlist**. If created successfully, then LEFT click one more time on **Examine Netlist**; whereby a notepad file opens. Everything will seem fine, except that we do not know what node names we are using. We can exit this window and proceed to name our nodes in the Schematics program.

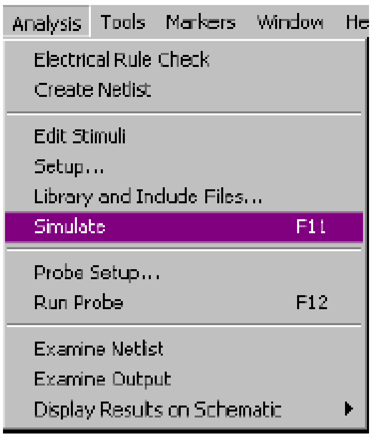


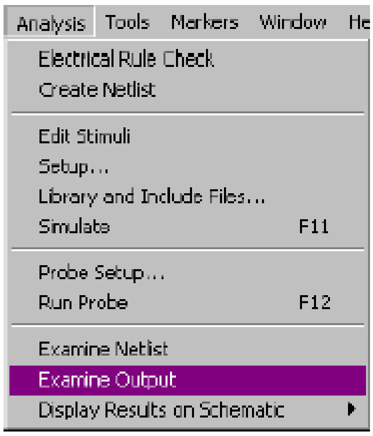


b. Double click on the LEFT mouse button on any wire you need to name. You will see a dialog box with **Set Attribute Value pop** out. Enter a name for this value, and the name of the wire here will become the name of the node. Make sure you leave no spaces between any name labels you want to assign. c. Another method is to use bubble. We get a part called **Bubble** from the **Get new part browser** and place it in the circuit. The bubble part usually comes with an attaching wire, if it does not you can draw a small wire first and attach the head of the bubble to the wire. You can then name the bubble by LEFT clicking on the graphic and labeling the name into the **Set Attribute Value** dialog box.

d. After you have done the naming, now perform **Create Netlist** and **Examine Netlist** again. It should display node names and locations in your circuit.

(4) In this step we would like to simulate the circuit. Again using your LEFT mouse button, click on the **Analysis menu bar**, and select **Simulate**. The Pspice simulation window will appear and the window will display the words **Bias point calculated**, signifying the node voltage analysis is complete. When the node voltage analysis is complete, LEFT click on the Schematics window to bring it to the front. You can see that the IPROBE and the VIEWPOINT parts display results of the node voltage analysis. These same results are placed in the output file. To examine the contents of the output file, select **Analysis** from the menu and then select **Examine Output**. The notepad program will display the output file contents.





TRANSIENT ANALYSIS:

(1) Transient analysis is used to look at waveforms versus time. Waveforms are displayed, as you would see them displayed on an oscilloscope screen. Sources used with the transient analysis include:

a. VSIN, ISIN: sinusoidal voltage or current source. Typical voltage waveform:). *V(t) = 5sin(2000t+30).*

b. VEXP, IEXP: can be used to create an exponential waveform. Typical current waveform: *I(t)=5(-exp(t/c))*.

c. VPULSE, IPULSE: pulse waveform, used to create square waveforms.

d. VPWL, IPWL: used to create an arbitrary waveform made up of straight lines.

e. VSFFM, ISSFM: used to create a frequency-modulated sine wave.

f. Vsq: a square wave voltage source. This source uses the pulsed voltage source to make a square wave. It is a special case of VPULSE.

g. Vtri: a triangle wave source. This source uses the pulsed voltage source to make a triangle wave. It is also a special case of VPULSE.

h. VRAMP: a saw tooth voltage source. This source uses the pulsed voltage source to make a saw tooth wave. It is also a special case of VPULSE.

(2) STEP CEILING. One of the features of the Transient Analysis that causes confusion is the step ceiling argument. Suppose we want to simulate a sinusoidal source. We expect to see a sinusoidal curve in PSpice. However, when PSpice runs a Transient Analysis, it solves differential equations to find voltages and currents versus time. The time between simulation points chosen in Pspice is chose as large as possible to keep the simulation error below a specified minimum. Due to he large steps, graphs don't come out looking sinusoidal. If we decrease the time between points we will see that all the points lie on a sinusoidal curve. Pspice provides a **Step Ceiling** argument when doing Transient Analysis; by reducing the **Step Ceiling**, the maximum time between simulation points, we increase the number of simulation points.

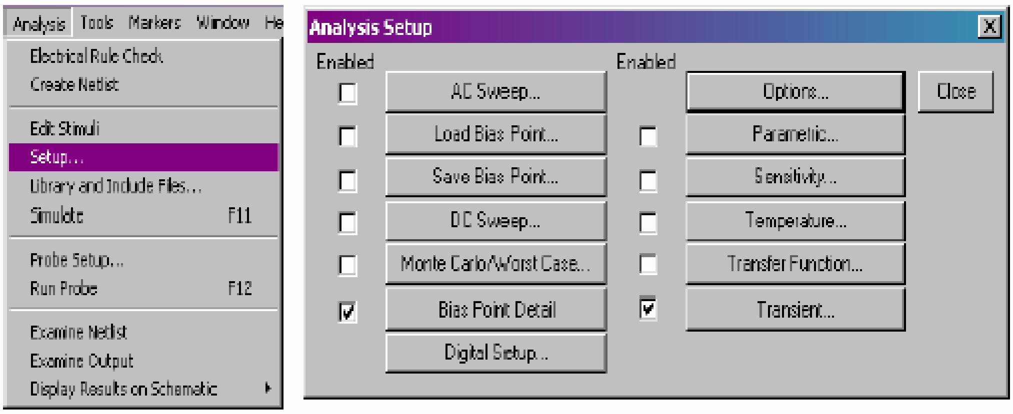
(3) CONVERGENCE. Another issue that may be encountered while using Transient Analysis is a convergence problem. During simulating a differential equation, PSpice calculates a data point and associates an error with the calculation. If the error is larger than the specified maximum, Pspice will reduce the time step until the error is within acceptable limits. This reduction will keep on happening up until the limit on the number of times Pspice is allowed to reduce the time steps will be reached. After this limit is reached, Pspice announces that the simulation has failed to convergence. Aside from that error message, another indication the simulation fails is that the time step used is very small. To get enough information on your simulation results, look into the **Examine Output** file from the **Analysis menu bar**. There are three ways to prevent convergence problems in your simulations:

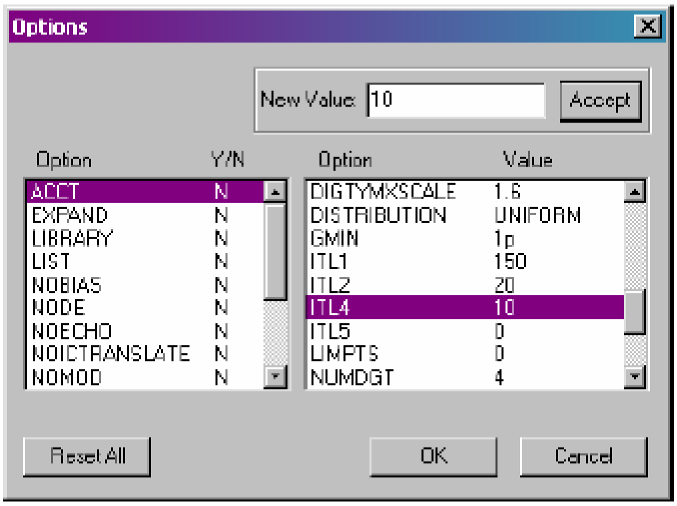
a. *Reduce the final time*. When you set up Transient Analysis, you must specify the final time of the simulation. If the final time is a large number, the initial time step is large and will yield a large amount of error. If the step ceiling is not specified, Pspice must make a guess for an appropriate value for the time step. It usually does this by dividing the final time by some number to get into an acceptable error range.

b. *Specify a small step ceiling value*. If you do not wish to change the final step value, make sure you specify a value for the step ceiling. The step ceiling is the largest value allowed for the time step. If the error is too large, Pspice will reduce the time step. Since you will start with a small value for your step ceiling, after a number of reductions, the step size will become small enough to make the error acceptable. If,

after making the step ceiling smaller the simulations fail to converge, reduce the step ceiling value even more. This will increase the simulation time as a result.

c. *Increase the number of time step reductions*. You can also increase the limit on the number of times the time step gets reduced. If the two methods above fail, this would be one way to solve the convergence issue. You first LEFT click on the **Analysis menu bar**, then select **Setup** from the list. Then you select the **Options** button and scroll down until you see ITL4=10. LEFT click on the text ITL=10 and change the value to 40 or 50, and click the **Accept** button to accept the number.





CAPACITOR CIRCUIT WITH INITIAL CONDITIONS:

We will observe the transient response of a capacitor with specified initial conditions of 5 volts and a circuit switch constrained to close at t = 1 ms. Notice the plus sign on the capacitor, make sure the sign is oriented as shown in the figure. The initial conditions on the capacitor is not a displayed attribute, therefore we must edit all the attributes of the capacitor. To do this we start by:

(1) Click on the LEFT mouse button on the capacitor graphic to select the part. When the graphic is highlighted in red, we select **Edit** and then **Attributes** from the Schematics menu bar. The dialog box for **C1** will appear (the default name of the capacitor). This box shows all the characteristics of the capacitor.

a. VALUE: is the value of the capacitor in farads. The default value is 1 microfarad.

b. IC: initial condition of the capacitor in volts. The default value is 0 volts.

c. TEMPLATE: is the line used in the Schematics used to create the Netlist for the capacitor. The value cannot be changed.

d. WORKING VOLTAGE: is for documentation purposes only. It has no effect on simulations and is provided to allow the user to display working voltages on Schematics.

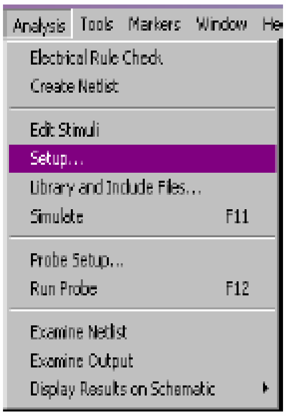
e. REFDES: illustrates the name of the capacitor. In our case we use the default name **C1**.

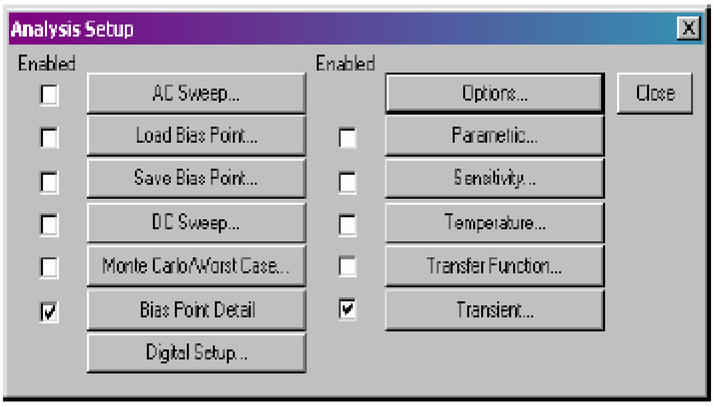
(2) To get the capacitor to an initial value of 5 volts, we LEFT click the mouse button on the text **IC =** and enter the number 5 in the **Value** box. Click the **Save Attr** button to save your changes.

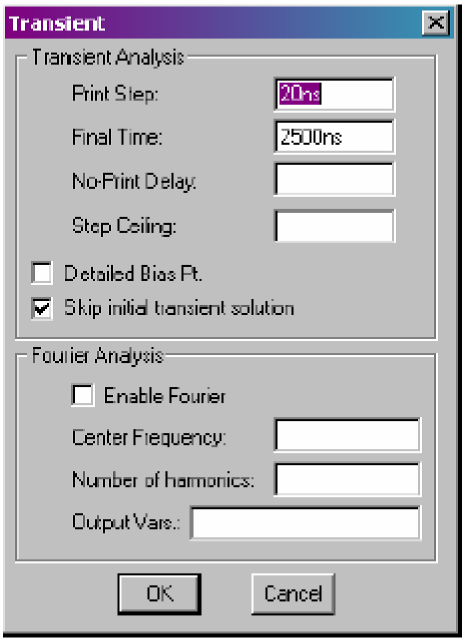
(3) We also would like to display the capacitor characteristics, so we LEFT click on the **Change display** button. To do this chooses the text **Both name and value**. Then click the **OK** button to accept the values, this returns you to the Schematics window which now displays condition and component attributes.

(4) After selecting a suitable switch for your circuit, we want to change the switch so that it closes at t = 1 ms. Double click the LEFT mouse button on the text t = 0. The **Set Attribute** dialog box appears. Now enter text t = 1 ms and click the **OK** button.

(5) Now click on the **Analysis** menu, select **Setup** and then click the **Transient** button to obtain a Transient analysis dialog box. The print setup is used for printing the text output. Every print second, the print part in the program will print out the specified values into the output file. We will be using probe to view the results of the Transient analysis, so this parameter is not important. We also specify the **Final time**, which is the length of the simulation. The **Step ceiling** is also important; to keep the simulation times at a minimum leave this value blank.







(6) The time constant of our circuit is **1 ms**, and after **5 times constants** the circuit should reach a steady state status. So we will set the simulation run for 5 time constants. However, to let the capacitor transient run for 5 ms we need a total simulation time of **6 ms**. And if we would like to see at least 500 points during the capacitor transient, we would have to set the **Step ceiling** to 5 ms / 500 which equals 0.01 ms.

(7) In the Transient dialog box there is another box labeled Skip initial transient solutions. If this box is checked PSpice will use the initial conditions specified in the circuit for the transient run. If the box is not checked, PSpice will calculate the initial conditions from the circuit. It assumes that all capacitors are open circuits and all inductors are short circuits.

(8) Click the **OK** button to accept the settings we put in, and the click the **Close** button to return to the Schematics window.

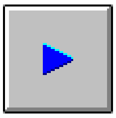
(9) To run simulation. Select **Analysis** and then select **Simulate** form the Schematics menu bar. The PSpice simulation window will appear. When the simulation is complete, probe will automatically run. Add the trace **V(Vr)** by selecting **Trace** and then **Add** from the probe menu bar.

THE ON SCREEN WINDOW OF THE COM3LAB COURESE

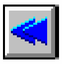
Introduction

COM3LAB courses are very easy to use, due to their uniform structure and Windows-based user environment.

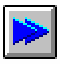
Like a book, a COM3LAB course is divided into chapters and pages. Some of the pages contain several processing steps.
Buttons for selecting the page to be displayed are positioned in the bottom right-hand corner of the COM3LAB window.



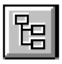
Click on this button to confirm that you have read the text, viewed the pictures or carried out the steps required by the program; continue takes you to the next processing step.



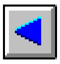
Click on this button to display the previous page.



Click on this button to stop processing the current page and display the next page.

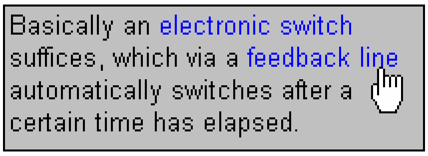


Click on this button to call up a course overview from where individual pages can be selected.



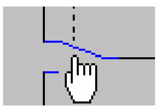
Click on this button to repeat a page sequence.

The work area of this COM3LAB course is divided into a descriptive area and a display area. The left half of the window in the descriptive area contains text on the current learning step. Many of these texts contain hotwords. A hotword is always displayed in blue and can be clicked on:

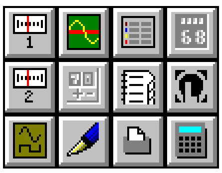


After a hotword is clicked on, an explanatory box describing the related item in closer detail appears in the display area. Click on every hotword you encounter to obtain full information on the related items.

The display area is located in the right half of the program window. It contains photographs and diagrams concerning the topic on that page. Some display areas contain blue colored or bordered hotspots, which you must click on just like the hotwords to obtain a description. You should also make ample use of this facility to obtain detailed information:



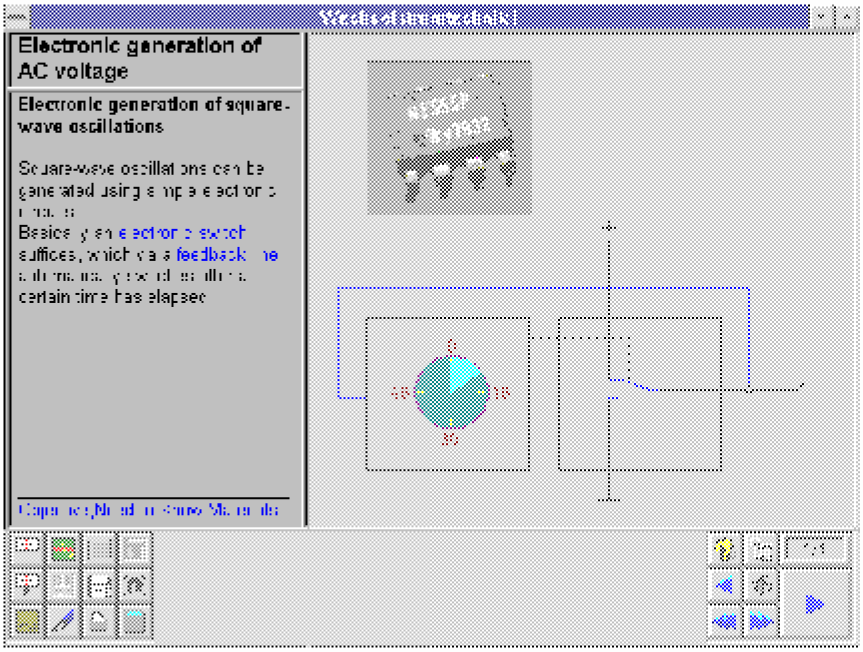
The tool box is located in the bottom left-hand corner of the COM3LAB window. If you require measurement instruments or other accessories for this experiment, click on the corresponding buttons.



The program messages of the COM3LAB course are displayed in the lower middle section. These can include prompts to carry out various operations, responses to your entries, help messages etc. Program messages can be several pages long, in which case you can click on a tag in the bottom right-hand corner to view the next page.

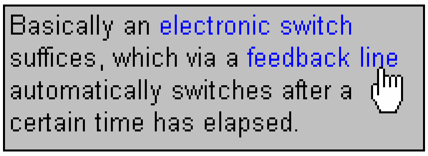
Descriptive area

The descriptive area of a page provides a brief explanation of the topic dealt with on that page. To obtain further details on this topic, click on each of the blue colored hotspots.



Hotword

The blue colored words in the descriptive area are termed hotwords.

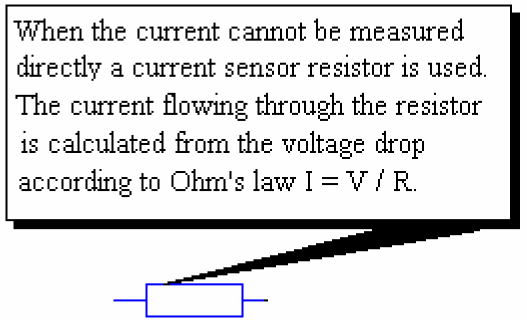


Clicking on a hotword calls up an explanatory box containing further details on the related item. If this item is displayed in the display area, the explanatory box usually carries a corresponding pointer. Some explanatory boxes contain further hotwords.
When you move the mouse cursor over a hotword, the cursor changes into a white hand to indicate that this text block can be activated.

Explanatory box

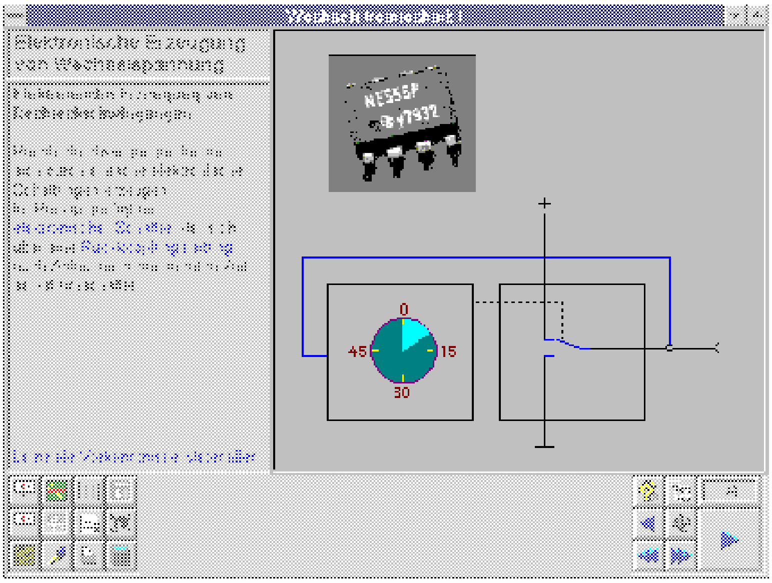
Explanatory boxes appear in the display area when you click on a hotword. They provide a detailed explanation of the related item and sometimes contain further hotwords.

The black pointer, on many explanatory boxes, points to the object concerned in the display area.



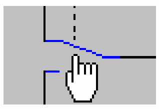
Display area

The display area comprises the large section on the right-hand side of the screen and contains illustrations concerning the current topic. The notebook appears in the display area of some pages.



Hotspot

Like a hotword, hotspots allow detailed information to be called up. Hotspots are blue or blue bordered objects in the display area.



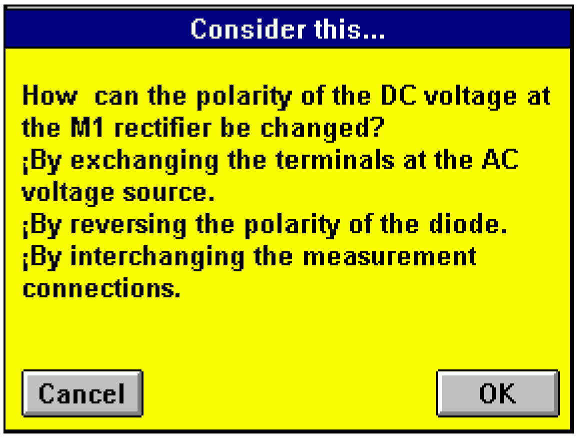
Clicking on a hotspot calls up an explanatory box containing further details on the related item. If this item is represented in the display area, the explanatory box usually carries a corresponding pointer. Some explanatory boxes contain further hotspots.
When you move the mouse cursor over a hotspot, the cursor changes into a white hand to indicate that this object can be activated.

Program message

The program messages of the COM3LAB course are displayed in the program window in the lower middle section. These can include prompts to carry out various operations, responses to your entries, help messages etc. Program messages can be several pages long, in which case you can click on a tag in the bottom right-hand corner to view the next page.

Question box

While the experiments are being performed, yellow question boxes occasionally appear on the screen in the following form:



Don't worry: These questions are not intended to examine your depth of knowledge, rather, they are meant to support a more accurate observation and evaluation of the experiment procedure.

Depending on the topic involved, there is either just one or more correct answers to a question. Correspondingly there are question boxes with a single choice or multiple choices, and these boxes are identical except for the type of selection element involved.

Due to the high similarity between these two types of question boxes, a complete explanation of the single-choice question boxes is provided here first (see the illustration above), then followed by notes on the comparative differences with multiple-choice question boxes (see the illustration below).

Single-choice question boxes

This question box usually contains the first part of a sentence for which several possible supplements have been proposed. Your task is to formulate a correct statement by choosing the suitable supplement.

The choice buttons are on the left beside the proposed sentence supplements, indicating which alternative which has been selected.
When a sentence supplement is selected, the corresponding choice button is marked with a dot.
Empty choice buttons indicate that the corresponding sentence supplements have not been selected.
When a question box first appears, all choices are unmarked to allow you to freely decide which of the possibilities is correct. Once you have made your decision, simply click on the corresponding choice with the mouse.

In any single-choice question box, you can only select one of the available choices by clicking on it, whereupon all other choices are deselected automatically.

After you have selected a choice, confirm your selection by clicking on "OK".

The program message which subsequently appears shows you whether your answer is correct. If not, you are given a further opportunity to answer the question.
After the question has been answered correctly, the corresponding observation or conclusion is usually entered in the appropriate page of the notebook. Even while a question box is displayed, the notebook allows you to view notes on different experiments which have already been conducted.

However, if you switchover to another page in the program window while a question box is displayed, the notebook entry remains incomplete. In this case, you should work through the whole page again by clicking on repeat.
If the question box covers a part of the screen you want to view, you can move it to a different position by

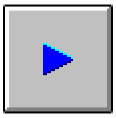
- positioning the mouse cursor on the header of the question box,
- holding the left mouse button down,
- moving the box to the desired position and then releasing the mouse button.

Multiple choices question boxes

Multiple-choice question boxes are essentially similar to single-choice question boxes. Instead of the choice buttons though, they contain rectangular check-boxes which are marked in the active state. As opposed to the buttons in the single-choice question boxes, several check-boxes can be active here at the same time.

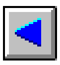
NAVIGATING WITH COM3LAB COURSES

Continue



The continue key allows you to proceed through a program page step-by-step. It is used to acknowledge program messages and entries, start graphic sequences, confirm circuit configurations etc.
Once all the steps in a page have been completed, the continue key brings you to the next page.

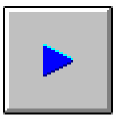
Repeat



Most pages change as their steps are processed. To restore a page to its initial state, click on repeat.
Also use this button if you want to work again on a page which you have already processed.

Change Page

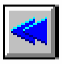
The COM3LAB user guidance system provides three possibilities of changing a currently displayed page.

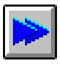


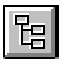
Normally, you will click on continue to proceed after having completed a step. After you have completed all the steps, the next page is called up automatically.

Note: some pages are not divided into individual steps. In such cases, the next page is

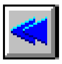
called up immediately.


Click on previous page to call up the previous page.


Click on next page to stop processing the current page and display the next one.

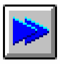

Click on course overview to obtain an overview of all pages. From this display, you can directly access any page.

Previous Page



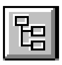
Click on this button to exit the current page and return to the previous one.
Note: If the previous page was already fully processed, it will appear in its final state, i.e. the processing sequence will not be commenced again. If you want to process such a page once more, you must also click on repeat.
In contrast, pages processed only partially or not at all are started from the beginning.

Next Page



Click on this button to leave the current page and jump to the next one.
Note: If the next page was already fully processed, it will appear in its final state, i.e. the processing sequence will not be commenced again. If you want to process such a page once more, you must also click on repeat.
In contrast, pages processed only partially or not at all are started from the beginning.

Course Overview



Click on this button to display the course overview:

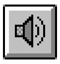
Each of the boxes in this overview relates to a page or learning step in the user guidance system. Pages belonging to the same chapter are listed together in a row of boxes. The topic handled by the chapter is listed above this row of boxes.
The state of each box indicates the corresponding stage you have reached in the COM3LAB course:

- Pages not yet processed are indicated by a blank box.
- Pages already processed are indicated by a marked box.
- The page from which you called up the course overview is indicated by a black box.

For you to retain a clear picture of all these boxes, a status line at the lower edge of the course overview indicates the topic handled by the page over whose box the mouse cursor is currently positioned.
Simply click on this box to call up the page.
Clicking on return brings you back to the learning step from which you called up the course overview.

Click on exit to exit the COM3LAB course (see Exit the COM3LAB course).

Audio



If you click on the button with the loudspeaker symbol, the COM3LAB texts are read aloud. Of course, this happens only if your PC is equipped with a suitable sound card and a loudspeaker or headphone connection.
Note: If you want to repeat the most recent spoken text, deactivate and activate the loudspeaker symbol once again.

Exit The COM3LAB Course

To finish work on the COM3LAB properly, carry out the following sequence of steps:

1. First you should exit the COM3LAB course. For this, click on course overview. In the course overview, click on exit and answer the subsequent prompt with yes. One of three equivalent procedures still needs to be carried out to exit any Windows program:
    - Click on the system menu box (the small box with the horizontal bar in the top, left-hand corner of the COM3LAB window) to call up the system menu. From this menu, select the close command to close the main program window and exit the program.

    - You can also exit the program by double clicking on the system menu box.
    - Alternatively, you can exit the program by holding the alt key down and pressing the F4 function key.

2. Take the CD-ROM of the COM3LAB course out of its drive and put it into its storage sleeve.
3. Exit Windows after exiting the COM3LAB course.

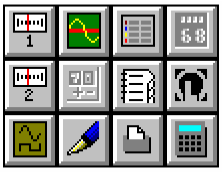
    - Windows 3.1 and 3.11: Click on the icon or window of the program manager, select the close command in the system menu and answer the subsequent prompt by clicking on OK.
    - Windows 95: Click on the start button and then on exit. The default option in the subsequent prompt should be "exit Windows".
      Click on OK.

4. Once the prompt to switch the computer off (Windows 95) or the DOS prompt appears, you can safely switch the computer off.
5. Turn off the COM3LAB master unit.
6. If required, you can then disconnect the data interface from the computer to the COM3LAB and unplug its power supply unit.
7. Remove the experiment board from the master unit and store it, together with the CD-ROM, in its case. The power supply unit of the COM3LAB can be stored in the master unit, after which you should replace the housing cover.

THE COM3LAB COURSE TOOL

Tool Box

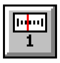
All the auxiliary programs of a COM3LAB course can be called up from the tool box in the bottom left-hand corner of the program window.



The toolbox contains the buttons for the measurement instruments, notebook, calculator, word processing and printer. The Copyright message is also activated from the toolbox.

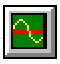
Measurement Instruments

The following measuring instruments are provided by COM3LAB:



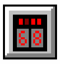
Two multimeters for measuring direct/alternating voltages and currents as well as resistances,

One function generator with square-wave, triangular and sinusoidal curve shapes as well as an adjustable, direct voltage source.

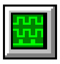


One oscilloscope with two signal inputs and digital storage.

One frequency counter can also be used as an event counter.



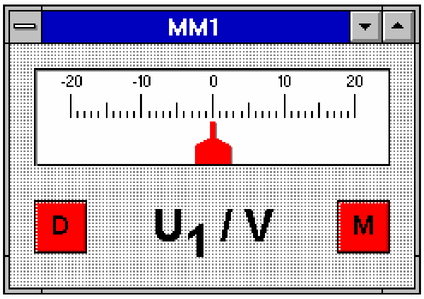
One 9-bit level indicator with binary, octal, decimal and hexadecimal displays.



One 9-bit digital analyzer with a freely selectable trigger word and 2048-word memory.

Multimeter

Click on this button to call up a window showing the multimeter display and control elements. Click on the button again to close the window. The windows for multimeters 1 and 2 are used in a completely identical manner.



The right-hand button M switches through the operating modes of the multimeter:

- direct voltage measurement,
- alternating voltage measurement,
- direct current measurement,
- alternating current measurement,
- resistance measurement.

The functions of this button are identical to those of the button on the master unit of the COM3LAB.

The left-hand button D switches through the display modes of the multimeter:

- analog,
- digital,
- digital with analog trend display.

The digital display in the multimeter window is identical to the display on the COM3LAB hardware. The multimeter window can be adjusted in terms of size and position, maximized to full-screen and minimized to an icon just like any other Windows application.

Note: If you position a multimeter window completely outside the main window of the COM3LAB course, it is not closed automatically by the program when a new page is called up. If you close such a window and open it again later, it is displayed again with the same size and position originally selected by you.

A multimeter window which was not changed or which overlaps the main window of the COM3LAB course is usually closed on the selection of a new page. The next time this multimeter window is invoked, it appears at a position determined by the program. If you close this window and open it again, it appears at its original position.

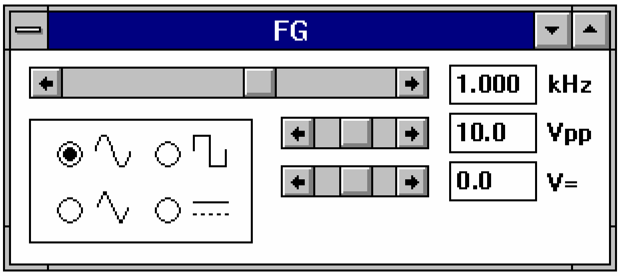
Multimeter specifications:
3½-digit display
Voltage measurement range: 2 V and 20 V,
autoranging*
Current measurement range: 0.2 A and 2 A,

autoranging*
Resistance measurement range: 2 kOhm,
20 kOhm, 200 kOhm, 2 MOhm, autoranging*

* Autoranging and switchovers between operating modes can be disabled by the COM3LAB course.

Function Generator

Clicking on this switch calls up a window displaying the various functions and control elements of the function generator. Clicking on the switch again closes this window.



The selector switches allow you to choose the signal shape (function) to be generated:

- sinusoidal,
- triangular,
- square-wave,
- direct voltage.

The usage of the selector switches is completely identical to that of the FUNCTION key on the COM3LAB hardware.

The number of control elements depends on the selected function. All control elements are visible, for example, when a square-wave signal is generated:

- frequency f,
- signal deviation Upp,
- zero offset U=,
- pulse-duty ratio.

A control element can be operated as follows:

- Drag the sliding knob with the mouse until the correct magnitude of the value is achieved.
- Click on the arrow keys to the left and right of the sliding knob to change the value indicated at the last position.
- The value can be entered numerically in the digital display.

The usage of these control elements is completely identical to that of the MODE key and the control dial on the COM3LAB hardware. However, the advantage of the screen display is that all parameters of a function are visible at the same time.

The window of the function generator can be adjusted in terms of size and position, maximized to full-screen and minimized to an
icon just like any other Windows application. The size of the control elements always remains the same though.

Note: If you position a function generator window completely outside the main window of the COM3LAB course, it is not closed automatically by the program when a new page is called up. If you close such a window and open it again later, it is displayed again with the same size and position that you originally selected.
A function generator window which was not changed or which overlaps the main window of the COM3LAB course is usually closed on the selection of a new page. The next time this function generator window is invoked, it appears at a position determined by the program. If you close this window and open it again, it appears at its original position.

Function generator specifications:
Terminals: Signal output OUT, synchronization signal SYNC and ground.
In some COM3LAB courses, the function generator is also used as an adjustable source of direct or alternating voltages for electrical circuits. In this case, the signal output is connected with the relevant experiment panel using the plug connector of the experiment board. The function generator also has an inverted signal output for generating positive and negative direct voltages symmetric about zero, opposing alternating voltages and higher direct voltages. However, this output can only be connected to certain experiment panels using the plug connector of the experiment board.

Output voltage: max. ±10 V*
Output current: max. 250 mA
Functions: sinusoidal, triangular, square-wave, direct voltage
Frequency range: 0.5 Hz to 100 kHz, frequency-dependent resolution
Signal deviation: max. 20 V*
Zero offset: -10 V to +10 V*
Pulse-duty ratio: 5% to 95% in 5% steps

* If the sum of the signal deviation and the zero offset exceeds +- 10V, the signal is clipped and the display on the COM3LAB hardware flashes.

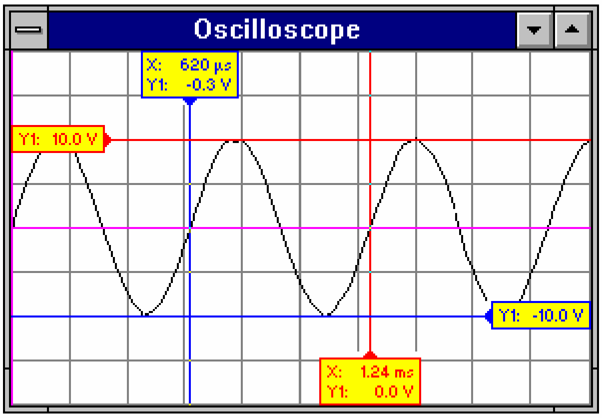
The COM3LAB can lock all function selectors and parameters against any attempts at modification. If such an attempt is made, a key symbol appears in the display of the COM3LAB hardware.

Oscilloscope

Display screen of the oscilloscope

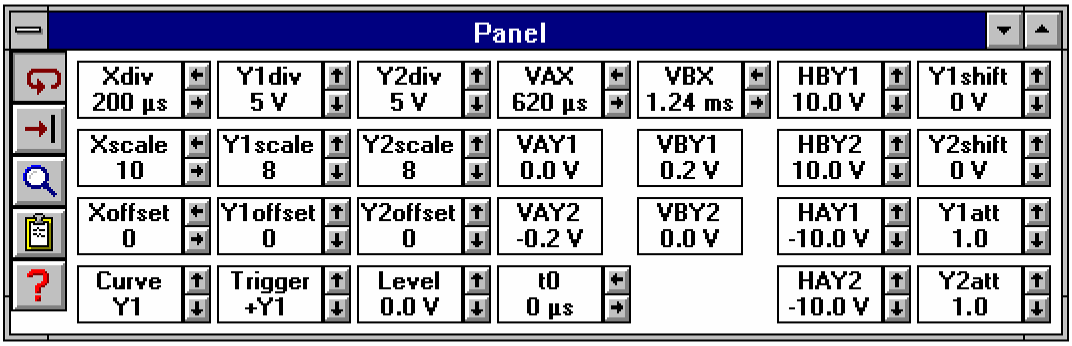
Panel of the oscilloscope

There are two windows for the oscilloscope: the display screen and the panel.



The display screen shows the time characteristic of the signal on one or both input channels of the oscilloscope.
The display screen has two vertical marker lines which can be dragged inwards from its left and right edges using the mouse. They indicate the signal voltages on the curves at the intersection points.

The display screen also has two horizontal marker lines which can be dragged inwards from its upper and lower edges using the mouse. They indicate the marker level corresponding to the scaling of one or both channels.



The panel allows the oscilloscope to be set. The display shows the entire panel with all its control elements:

- XDiv Sweep per division.
- XScale Zoom: number of horizontal grid lines (one complete sweep covers ten grid lines).

- XOffsetZoom: number of grid lines before the display range (XOffset = 0 when the full sweep period is displayed).
- Curve Display selection: Y1, Y2, Y1 and Y2, XY, Y1+Y2 (scaling of Y1), Y2+Y1 (scaling of Y2).
- Y1Div Signal sweep per division.
- Y1Scale Zoom: number of vertical grid lines (the full signal range covers eight grid lines).

- Y1Offset Zoom: number of grid lines before the display range (Y1Offset = 0 when the full measurement range is displayed).
- Trigger Trigger signal selection: OFF, +Y1, +Y2, +EXT, -Y1, -Y2, -EXT; + means rising edge, - means falling edge.
- Y2Div Signal deflection per division.
- Y2Scale Zoom: number of vertical grid lines (the full signal range covers eight grid lines).
- Y2Offset Zoom: number of grid lines before the display range (Y2Offset = 0 when the full measurement range is displayed).

- Level Level of the trigger threshold.
- VAX Position of vertical marker line A (this marker is initially positioned on the left edge of the screen).
- VAY1 Signal voltage of channel Y1 at the position of vertical marker line A.
- VAY2 Signal voltage of channel Y2 at the position of vertical marker line A.
- t_0 Zero offset of the display. The display can be offset in the positive or negative direction by one complete sweep period.

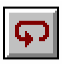
- VBX Position of the vertical marker line B (this marker is initially positioned on the right edge of the screen).
- VBY1 Signal voltage of channel Y1 at the position of vertical marker line B.
- VBY2 Signal voltage of channel Y2 at the position of vertical marker line B.
- HBY1 Level of horizontal marker line B (this marker line is initially positioned at the upper edge of the screen) with respect to the scaling of channel Y1.
- HBY2 Level of horizontal marker line B (this marker line is initially positioned at the upper edge of the screen) with respect to the scaling of channel Y2.

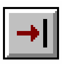
- HAY1 Level of horizontal marker line A (this marker line is initially positioned at the lower edge of the screen) with respect to the scaling of channel Y1.
- HAY2 Level of horizontal marker line A (this marker line is initially positioned at the lower edge of the screen) with respect to the scaling of channel Y2.
- Y1Shift Beam drift of channel Y1.
- Y2Shift Beam drift of channel Y2.
- Y1Att Attenuation of channel Y1. Negative attenuation factors can also generate inverted signal displays. This attenuation is taken into account when the signals are added.

- Y2Att Attenuation of channel Y2. Negative attenuation factors can also generate inverted signal displays. This attenuation is taken into account when the signals are added.

Note: In some cases, it is also possible to display just the required control elements.

The following buttons are arranged in vertical order at the left edge of the operating console:


Continuous recording on/off.


Take a snapshot.

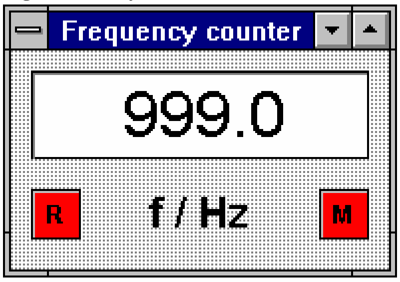

Zoom on/off. After the zoom is turned on, you can use the mouse cursor to square out an area which is then magnified.


Copies the data and the contents of the display area to the clipboard.

Calls up this Help topic directly.

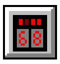
Frequency Counter

The frequency counter counts events from the inputs of the oscilloscope (Y1, Y2 or trigger) or the digital analyzer.



Button R resets the counter and starts a new counter cycle. Button M can be used to switch between frequency measurement (in two measurement ranges) and pulse counting.

Level Indicator



The level indicator shows the state of the nine data inputs of the digital analyzer.

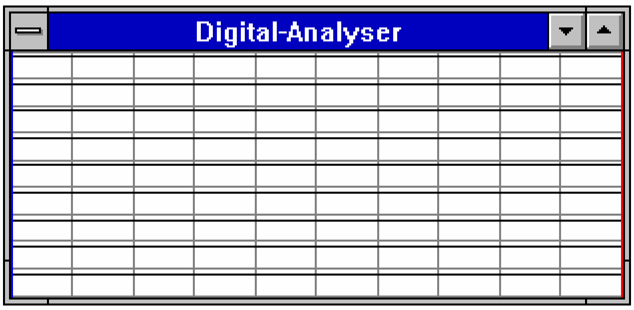
The display is binary on the LEDs and numeric in the digital field. An octal, decimal or hexadecimal base can be chosen for the numeric display.

Digital analyzer

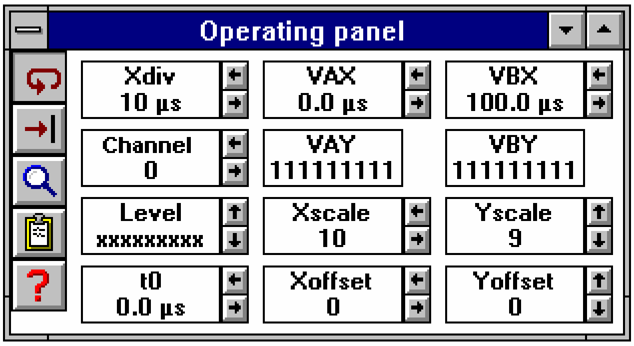
Display screen of the digital analyzer

Panel of the digital analyzer

The digital analyzer consists of two windows: the display screen and the panel.



The display screen shows the time characteristic of the logic level of the nine input channels of the digital analyzer. The display screen has two vertical marker lines which can be pulled inwards from its edges using the mouse. They indicate the data pattern in binary form.



The panel allows the digital analyzer to be set. The entire panel with all its control elements is displayed:

- XDiv Sweep per division.
- Channel Number of the channel which is modified with the trigger setting.
- Level Trigger word; every trigger bit can be set to 0, 1 or 'don't care' (x).
- t_0 Zero offset of the display. The display can be offset in the positive or negative direction by one complete sweep period.
- VAX Position of marker line A (this marker line is initially positioned at the left edge of the screen).
- VAY Contents of the data word at the position of marker line A.

- XScale Zoom: Number of horizontal grid lines (one complete sweep period covers ten grid lines).
- XOffsetZoom: Number of grid lines before the display range (XOffset = 0 when the full sweep period is displayed).
- VBX Position of marker line B (this marker line is initially positioned at the right edge of the screen).
- VBY Contents of the data word at the position of marker B.
- YScale Zoom: Number of displayed channels.

- YOffsetZoom: Number of channels not displayed (YOffset = 0 when all channels are displayed).

Note: In some cases, it is also possible to display just the required control elements.

The following buttons are arranged in vertical order at the left edge of the operating console:

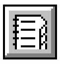
Continuous recording on/off.

Take a snapshot.

Zoom on/off. After the zoom is turned on, you can use the mouse cursor to square out an area which is then magnified.

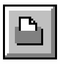
Copies the data and the contents of the display area to the clipboard.
Calls up this help topic directly.

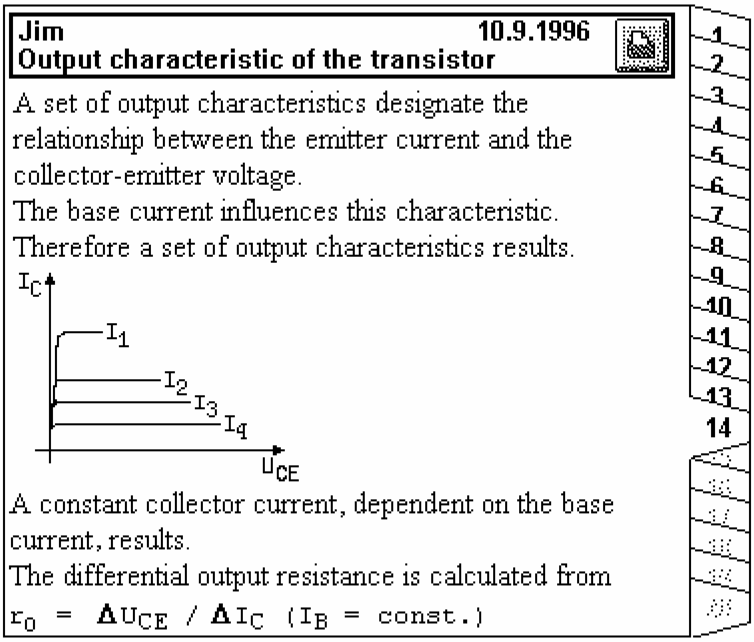
Notebook



The notebook contains findings and observations from previously processed chapters. While a chapter is being worked through, the corresponding notebook page is filled gradually with texts, tables and graphics. With every addition, the appropriate page of the notebook is displayed automatically.

To view the notebook at any other time, simply click on its button in the tool box. Click on this button again to close the notebook.

Click on  to output the contents of the open notebook page to a printer.



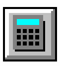
To refresh your memory, you can view notes on previously performed experiments by clicking on the corresponding page tabs (even while working on the current experiment).

Only the notebook pages of already completed chapters can be accessed. Clicking gray page tab numbers has no effect.

Word processing

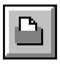
Windows Write can be invoked from the COM3LAB course for the purpose of storing text. This program is suitable for logging individual experiments, for example.
To close Windows Write, use the standard method or click again on the corresponding button in the COM3LAB course.

Calculator



The Windows calculator can be invoked from the COM3LAB course when calculations need to be performed.
Note: Calculated results can be copied to the clipboard and inserted into the tables of the COM3LAB course.
To close the calculator, use the standard method or again click on the corresponding button in the COM3LAB course.

Printer



The printer symbol in the main window of the COM3LAB course can be used to print out the current page or the contents of the clipboard.

The printing procedure is as follows:

1. First select what you want to print out.

    - To print the current page, click on this button and select the 'current page' option in the printer dialog box. Cascaded windows showing measurement instruments, the notebook and other Windows applications are not printed out!
    - To print out the current notebook page, click on the corresponding button in the notebook.
    - To print out the entire screen contents (including any other windows and the background), first press the print screen or print key on the keyboard. These copies the screen contents to the clipboard. Then click on the printer symbol and select the 'clipboard' option in the printer dialog box.

    - To print out a specific window, e.g. a multimeter display, first click on the title bar of that window. The window must be active! Then click on print screen. This copies the contents of the selected window to the clipboard. After that, click on the printer symbol and select the 'clipboard' option in the printer dialog box.

2. Enter the required number of copies in the 'copies' entry field.
3. Normally, the standard printer of the print manager is used. By pressing the 'settings' button, you can select a different printer. By pressing the next 'settings' button, you can modify the configuration of the selected printer.
4. Now click on the OK button.