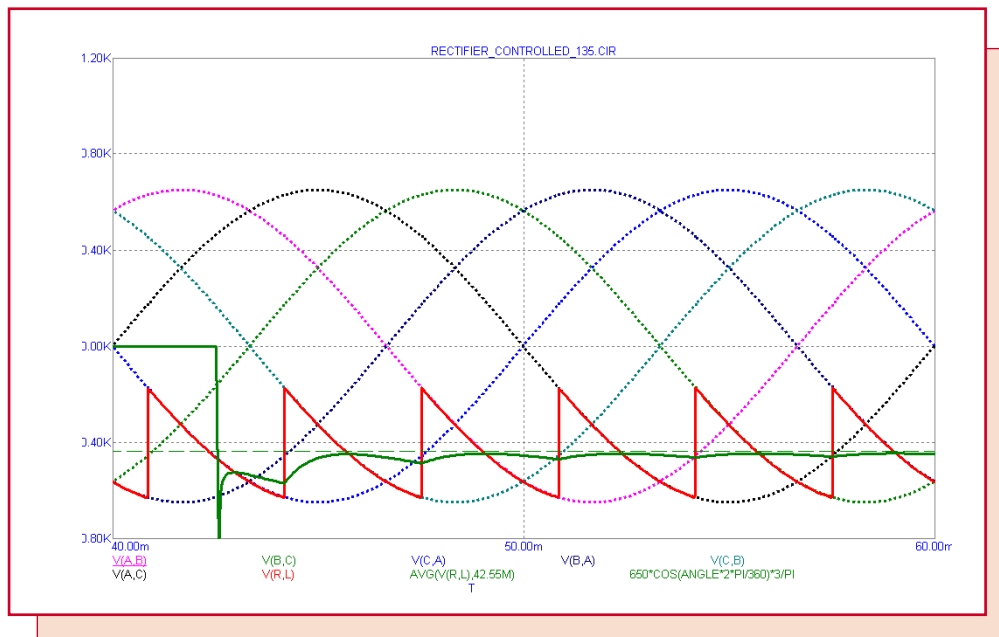


## Summer 2002 News



### Three Phase Rectifier/Inverter Circuits

Featuring:

- Using IBIS Models
- Six Pulse Diode Rectifiers
- Phase-controlled Six Pulse Rectifier/Inverter

---

## News In Preview

This newsletter's Q and A section describes how use symbolic variables to quickly make changes to the values of many components and how to recover your Component library files after a reinstallation overwrites them.

The first article describes a method for translating the output files from an IBIS2SPICE translator.

The second article describes a model for a three-phase six-pulse diode rectifier.

The third article describes a model for a three-phase six-pulse rectifier/inverter with thyristor control.

## Contents

News In Preview .....	2
Book Recommendations .....	3
Micro-Cap Questions and Answers .....	4
Easily Overlooked Features .....	5
Using IBIS Models .....	6
Six Pulse Diode Rectifiers .....	9
Phase-controlled Six Pulse Rectifier/Inverter .....	13
Product Sheet .....	16

---

## Book Recommendations

### General SPICE

- *Computer-Aided Circuit Analysis Using SPICE*, Walter Banzhaf, Prentice Hall 1989. ISBN# 0-13-162579-9
- *Macromodeling with SPICE*, Connelly and Choi, Prentice Hall 1992. ISBN# 0-13-544941-3
- *Inside SPICE-Overcoming the Obstacles of Circuit Simulation*, Ron Kielkowski, McGraw-Hill, First Edition, 1993. ISBN# 0-07-911525-X
- *The SPICE Book*, Andrei Vladimirescu, John Wiley & Sons, Inc., First Edition, 1994. ISBN# 0-471-60926-9

### MOSFET Modeling

- *MOSFET Models for SPICE Simulation, Including BSIM3v3 and BSIM4*, Wiley-Interscience, First Edition, ISBN# 0-471-39697-4

### VLSI Design

- *Introduction to VLSI Circuits and Systems*, John P. Uyemura, John Wiley & Sons Inc, First Edition, 2002 ISBN# 0-471-12704-3

### Micro-Cap - German

- *Schaltungen erfolgreich simulieren mit Micro-Cap V*, Walter Gunther, Franzis', First Edition, 1997. ISBN# 3-7723-4662-6

### Micro-Cap - Finnish

- *Elektroniikkasimulaattori*, Timo Haiko, Werner Soderstrom Osakeyhtio, 2002. ISBN# ISBN 951-0-25672-2

### Design

- *Microelectronic Circuits High Performance Audio Power Amplifiers*, Ben Duncan, Newnes, First Edition, 1996. ISBN# 0-7506-2629-1
- *Microelectronic Circuits.*, Adel Sedra, Kenneth Smith, Fourth Edition, Oxford, 1998

### High Power Electronics

- *Power Electronics*, Mohan, Undeland, Robbins, Second Edition, 1995. ISBN# 0-471-58408-8
- *Modern Power Electronics*, Trzynadlowski, 1998. ISBN# 0-471-15303-6

### Switched-Mode Power Supply Simulation

- *SMPS Simulation with SPICE 3*, Steven M. Sandler, McGraw Hill, First Edition, 1997. ISBN# 0-07-913227-8
- *Switch-Mode Power Supply SPICE Simulation Cookbook*, Christophe Basso, McGraw-Hill 2001. This book describes many of the SMPS models supplied with Micro-Cap.



---

## Micro-Cap Questions and Answers

**Question:** How do I set it so that a first group of capacitors has value X uf, a second group has value Y uf, and so on. This will let me only specify X, Y, and so on.

**Answer:** Use symbolic variables. Add two or more pieces of text to the schematic like this:

```
.DEFINE X 10n  
.DEFINE Y 50n  
...
```

Then set the VALUE attribute to X for the first group of capacitors, to Y for the second group and so on. Then you can easily edit the .DEFINE statements or even step the values of X, Y, ... in the analysis run.

The symbolic variables text can be placed directly on the schematic, in the text area of the schematic, or in the user file at Options menu / User Definitions so that the values are available to all circuits.

**Question:** I recently reinstalled Micro-Cap from the CD and now the changes I made to the Component library are gone. Is it possible to recover these changes as I spent a long time putting them in?

**Answer:** When you reinstall Micro-Cap, the existing Component library and Shape files (standard.cmp and standard.shp) will be overwritten unless you specifically deselect them in the Installation dialog box. If this has happened, you can restore the older versions since backup copies are made by the installer on the MC6 or MC7 folder as standard0.cmp and standard0.shp. If standard0.cmp and standard0.shp already exist due to a reinstallation, then the existing standard.cmp and standard.shp are renamed to standard1.cmp and standard1.shp.

The oldest files are those with a 0 in their name. The most recent files are those with the largest number in their name, and are the ones in use prior to the most recent reinstallation. It is these highest numbered files that you want to recover by renaming.

Simply delete or rename the existing standard.cmp and standard.shp files and then rename the standardN.cmp as standard.cmp and the standardN.shp as standard.shp, where N is the largest file number present. Do this from the desktop or DOS before running MC6/MC7.

---

## Easily Overlooked Features

This section is designed to highlight a few features per issue that may be overlooked because they are not made visually obvious with an toolbar button.

### Renaming Component Names

Ever create a schematic and, after numerous deletions and additions, find that the parts are not named sequentially like R1, R8, R11? If you are like most engineers, you may want to tidy up the schematic by renaming the parts. Other than manually changing the names is there another way? Yes there is. Select **Edit menu / Change / Rename Components** and Micro-Cap will rename the parts according to the usual naming convention; R for resistors, C for Capacitors, etc. It scans the parts from the upper left to the lower right by rows and renumbers all of the resistors a R1, R2, R3,... then all of the capacitors as C1, C2, C3... and so on for each part type.

### Connecting Nodes Together

To connect one node with another you normally drag between the nodes. Sometimes this is inconvenient, and occasionally impossible when the nodes are on different pages. What to do? Use text to connect the nodes together. The same text placed on two schematic nodes connects them together. It is as simple as that. Placing text on a node labels or names the node so that it can be referenced in an expression, such as when you want to print or plot V(OUT) or simply use it in another expression.

### Locating Parts in the Component Library

When you want to know if there are any thermistors in the library you could simply browse the Component menu and see if there are any included. An easier way is to use the Find Component command. This command can be found on the Component menu, but it is more readily invoked by SHIFT + CTRL + F. This invokes its dialog box from where you can search the entire Component library by field for any text. In this case you would type in "thermistor", then click on the All button to search all fields of the Component library, and then click OK. This produces a list of part names that have the text "thermistor" in one of their field names. The list itself can be placed in a text file for later use if you can simply select one of the part names from the list.

You use this command to locate all parts whose Name, Shape, Definition, and / or Comment field text matches your search string.



---

## Using IBIS Models

Micro-Cap 6 and 7 do not have an internal IBIS model, but IBIS models can be converted. This article describes how to translate the subcircuit files created by the IBIS2SPICE utility into subcircuits that Micro-Cap can use. This utility is available from Intusoft and can be downloaded from <http://www.intusoft.com/utilities.htm#Modeling>. It translates IBIS pin descriptions to subcircuit models in the Intusoft SPICE3 syntax. This article shows you how to convert these SPICE3 buffer models to Micro-Cap / PSpice compatible syntax.

### Caveats

First, be aware the IBIS2SPICE utility only produces a simple exponential that approximates the RAMP keyword and makes no attempt at all to produce the RISING WAVEFORM or FALLING WAVEFORM portions of the IBIS spec. The utility does an adequate job of modeling the output impedance of the IBIS buffer and a rough approximation to the switching time.

To illustrate we will use the MOT7410.IBS file which was downloaded from a Motorola web site. If you wish to use the file as a working example, it is available from our user group:

<http://groups.yahoo.com/group/micro-cap/files/mot7410.ibs>

Run IBISTOSPICE on the MOT7410.IBS file and select the A02 pin and the typical option. The program will create a file called MOT7410.LIB.

### What to change

To use the MOT7410.LIB file created by the utility, you will need to make several modifications.

#### 1) Switches:

The SMOD switch model syntax must be changed from this format:

```
.MODEL SMOD SW RON=.1M ROFF=1E15 VT=-1.2 VH=.1
```

to this format:

```
.MODEL SMOD VSWITCH RON=.1m ROFF=1E15 VON=-1.19 VOFF=-1.21
```

Change the device type from SW to VSWITCH. Use the same RON and ROFF parameters but replace VT and VH with VON and VOFF calculated as follows:

$$VON = VT + VH \text{ and } VOFF = VT - VH$$

#### 2) PWL G Tables

The voltage in current out pwl tables must be changed from this form:

```
APullDown %vd(1,2) %id(3,4) transfer
.model transfer pwl
+ fraction = FALSE, input_domain = 0.0, xy_array = [
...{table values}...
+ ]
```

---

to this form:

```
G1 3 4 table {v(1,2)} = (  
...{table values}...  
+ )
```

### 3) PWL E Tables

The voltage in voltage out pwl tables must be changed from this form:

```
APullDown %vd(1,2) %vd(3,4) transfer  
.model transfer pwl  
+ fraction = FALSE, input_domain = 0.0, xy_array = [  
...{table values}...  
+ ]
```

to this form:

```
E1 3 4 table {v(1,2)} = (  
...{table values}...  
+ )
```

Don't forget on the last line to change the bracket "]" to a parenthese ")".

Do this for the four PWL sources in the GND\_CLAMP, POWER\_CLAMP, PULL\_UP, and PULL\_DOWN subcircuits. You should be able to just cut and paste as all four PWL sources use the same syntax and node numbers.

### 4) Parameter definitions:

Change the two DEFINE statements from this format:

```
*DEFINE {RTF}= 60.868k ;0.73104/4.44969e-10  
*DEFINE {RTR}= 19.931k ;0.75231/1.49940e-10
```

to this format:

```
.PARAM RTF 60.868k  
.PARAM RTR 19.931k
```

### 5) B Devices:

Change the B3 and B4 device syntax from this format:

```
B3 300 850 I= V(830) > 1.2 ? 0 : V(300,850) / {RTR}  
B4 840 400 I= V(820) > 1.2 ? 0 : V(840,400) / {RTF}
```

to this format:

```
B3 300 850 I= IF(V(830) > 1.2,0,V(300,850) / {RTR})  
B4 840 400 I= IF(V(820) > 1.2,0,V(840,400) / {RTF})
```



Ordinarily, there are only two devices of this type, usually called B3 and B4.

Change the B1 and B2 syntax from this format:

```
B1 820 0 V=V(100) & V(500)
B2 830 0 V=V(500) & ~V(100)
```

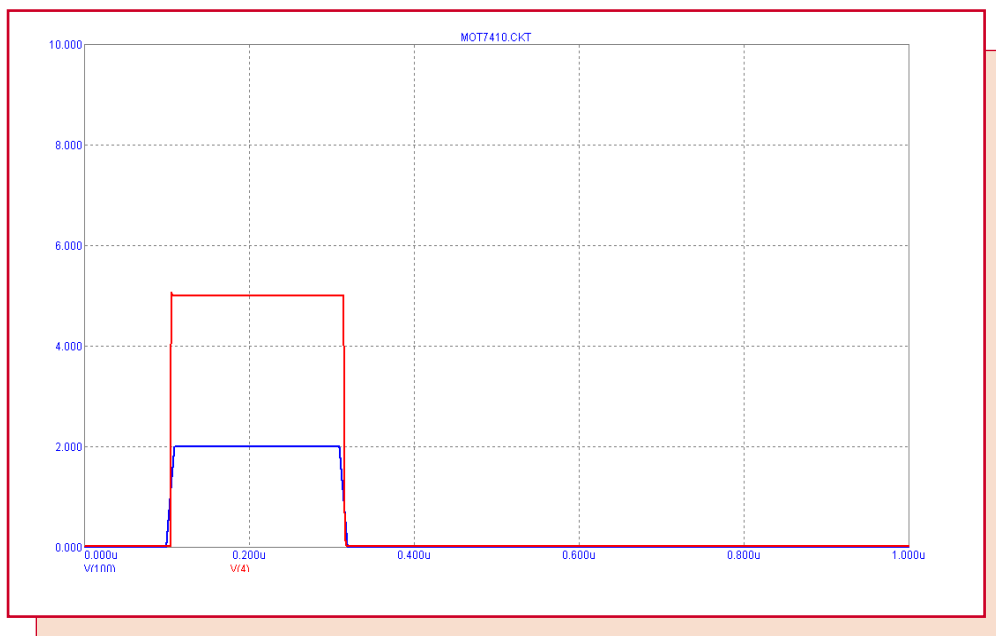
to this format:

```
B1 820 0 V={2*((V(100)>=1) AND (V(500)>=1))}
B2 830 0 V={2*((V(500)>=1) AND (V(100)<=1))}
```

Having made these changes to the MOT7410.LIB file, you are ready to begin using it in Micro-Cap. You can use it directly as a SPICE file by adding a pulse generator to drive it and a battery or two by adding these lines to the top of the file.

```
V1 100 0 PULSE 0 2 100n 10n 10n 200n 1u
R1 4 0 1E12
VCC 300 0 5
VEE 400 0 0
VEN 500 0 5
```

All you need to do now is to run transient analysis and plot the voltage on the input node, V(100), and the output node, V(4). Here is what the run looks like for a 1us analysis time:



**Fig. 1- Transient analysis of the converted M7410 IBIS file**

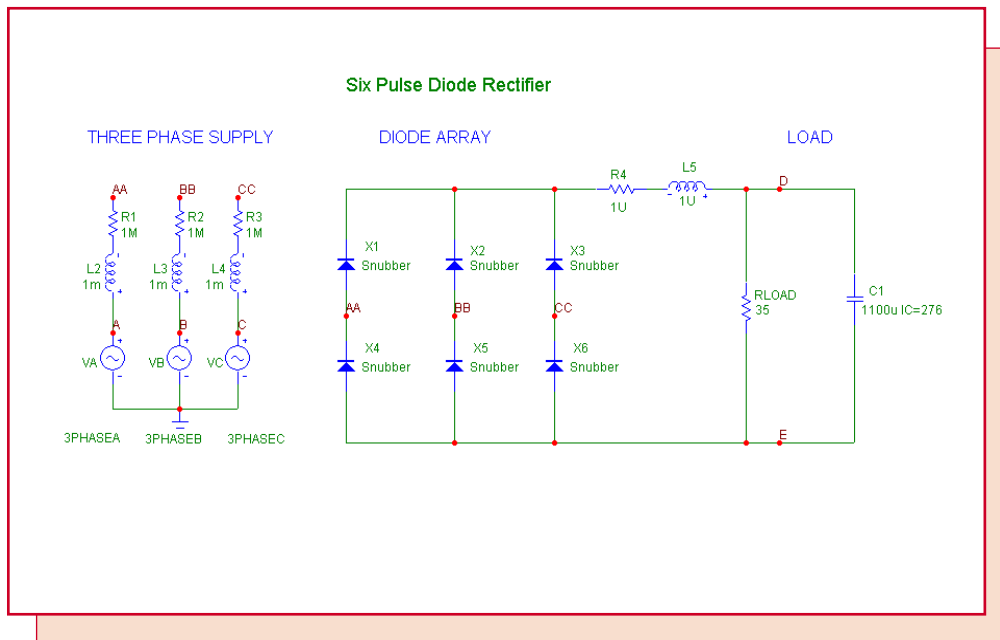
This completed circuit file with the modifications is also available in the user group File section at:

<http://groups.yahoo.com/group/micro-cap/files/mot7410.ckt>



## Six Pulse Diode Rectifiers

Six-pulse diode rectifiers are often used to produce a constant DC voltage starting with commonly available three phase line voltages. The general circuit is shown below:



**Fig. 2- Six-pulse diode rectifier**

This circuit is described in detail in the following references:

- 1) *Power Electronics*, by Mohan, Undeland, and Robbins Pages 103-120
- 2) *Modern Power Electronics*, by Andrzej Trzynadlowski, Page 126-129

Here is how to set up the circuit and the accompanying simulation.

First, create the three SINE sources to supply the three phases. Use the following model parameters for the Sine source.

### For all sources:

F0 = Frequency = 60

A = Peak Amplitude = 170

All other parameters = 0

### For each source

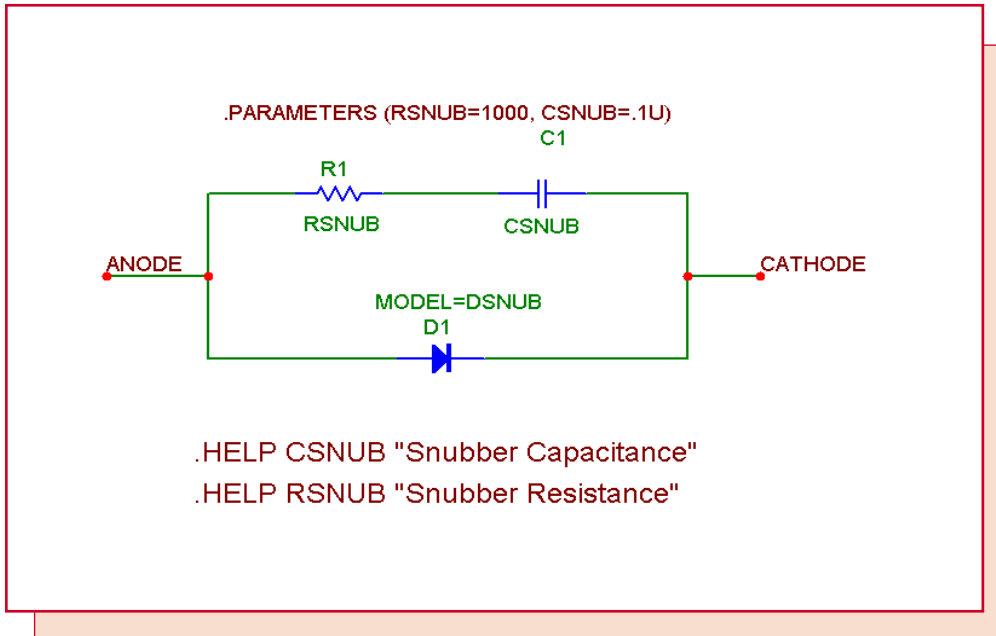
Source 1: PH = Phase added to the Sine argument = 0

Source 2: PH = Phase added to the Sine argument =  $-120 \cdot 360 / (2 \cdot \text{PI}) = -2.0944$  radians

Source 3: PH = Phase added to the Sine argument =  $-240 \cdot 360 / (2 \cdot \text{PI}) = -4.1888$  radians

This will produce a balanced set of 3 sine waves of amplitude 170v (120 RMS) and frequency 60 Hz, separated by the requisite 120 degrees.

The snubbing diodes are power diodes capable of handling the current and voltage with an energy dissipating R-C filter between the diode's two terminals. These filters provide protection to the circuit by eliminating fast transients and at the same time they frequently make the circuit more convergeable and thus easier to simulate. In this case we use a generic diode with an on resistance of 10m and infinite breakdown. Here is the schematic of the snubbing diode.



**Fig. 3- The snubber schematic**

The rectifier has six diodes arranged in two groups. The common cathode group consists of X1, X2, and X3, and the common anode group consists of X4, X5, and X6. One diode from each group is conducting at all times. For example, X1 and X5 fire together whenever the voltage  $V(A) - V(B)$  is greater than any other line-to-line voltage. Altogether there are six combinations of diode pairs that fire together to complete the cycle. Each pair fires for 60 degrees or 1/6 of the 16.667ms (60 hZ) period of the sinusoidal inputs.

The approximate average voltage across the output load,  $V(RLOAD)$ , is found from this formula:

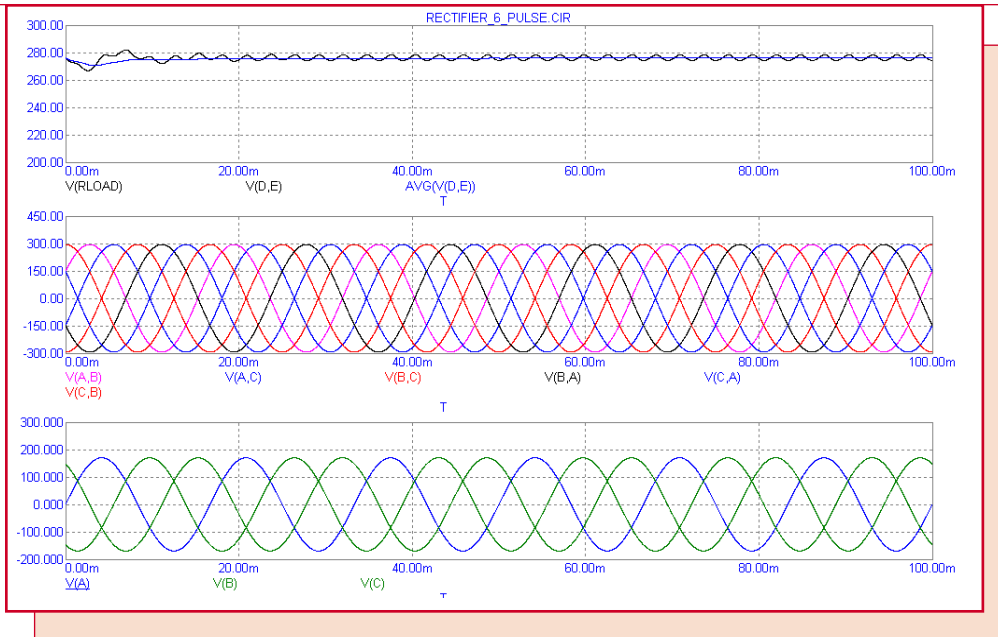
$$V_{OUT\_AVG} = V_{LLpeak} * 3 / \pi - V(X1) - V(X2)$$

The average DC output works out to about 95% of the peak line to line voltage - two diode drops. In this case:

$$V_{OUT\_AVG} = 294.45 * 3 / \pi - 1.03 - 1.03 = 279 \text{ volts}$$

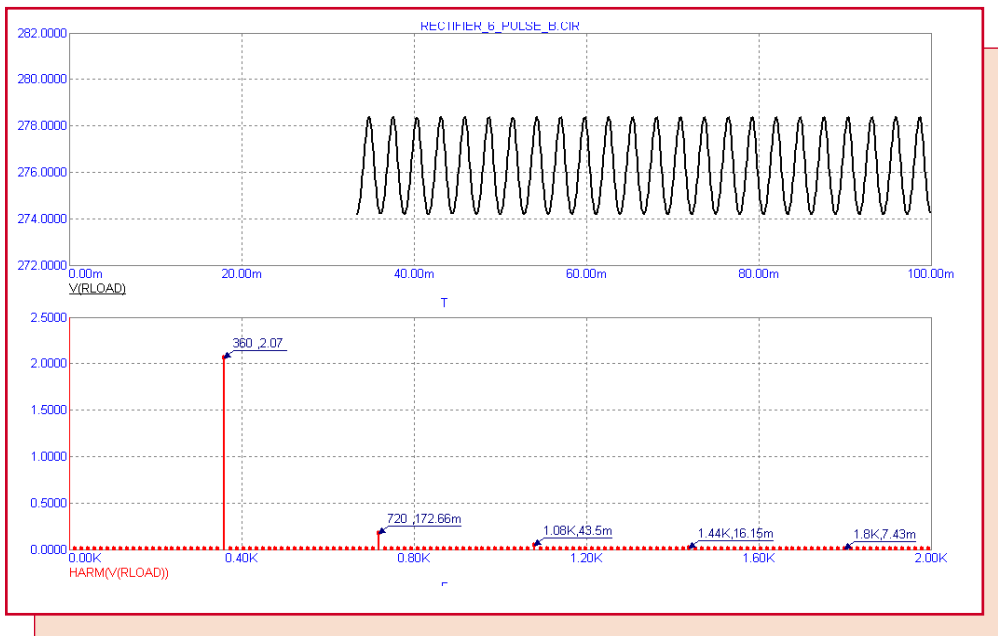
The ripple is best determined from a transient analysis simulation run.

A transient analysis of the circuit is shown in the next figure. The run time is 100msec.



**Fig. 4 The transient analysis run**

The average value of the load voltage from this run is actually about 276 volts. The ripple is about 4.2 volts in amplitude or 2.1 volts peak. The plot below shows the harmonic content found in the voltage across RLOAD. The principal components occur at multiples of 360 Hz due to the six-way commutation ( $6 \cdot 60\text{Hz}$ ).

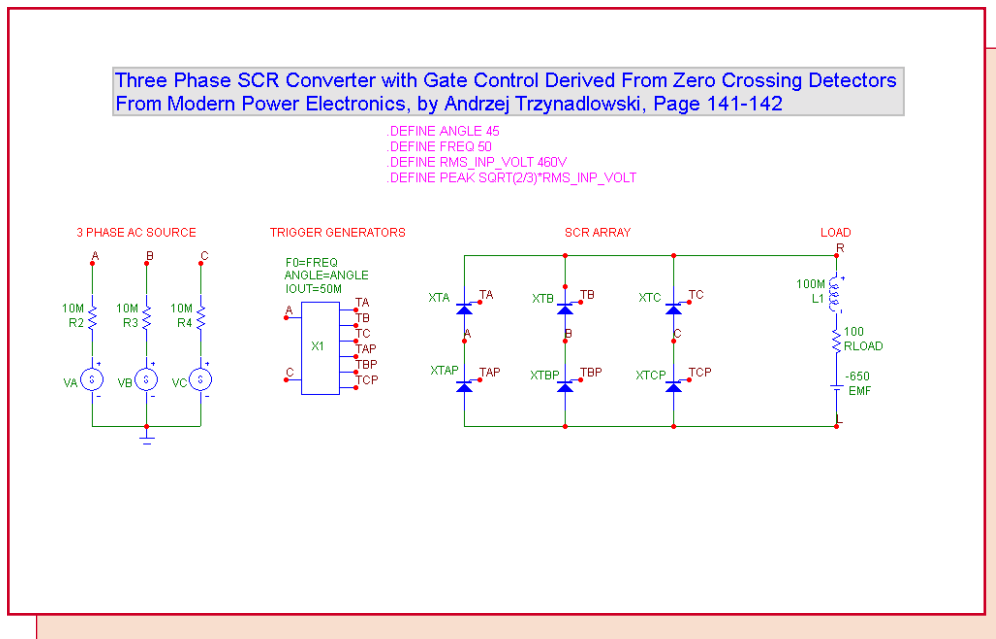


**Fig. 5 Harmonic content of V(RLOAD)**



## Phase-Controlled Six-Pulse Rectifiers

Six-pulse diode rectifiers work fine when you want a fixed DC voltage. When a controlled voltage is needed, a thyristor-fired phase-controlled rectifier may be preferred. Here is an example of such a beast.



**Fig. 6- Phase-controlled six-pulse rectifier**

In this type of converter the switching is delayed from the moment of natural commutation that would occur in a diode rectifier when the diodes are forward biased. This delay is called the firing angle and it can vary from 0 to 180 degrees. The delay reduces the average output voltage according to the following idealized formula:

$$V_{OUT(AVG)} = (3/\pi) * V_{PEAK} * \cos(\text{ANGLE})$$

Where  $V_{PEAK}$  is the maximum or peak value of the line-to-line voltage  $V(A,B)$ .

Several things are important to note:

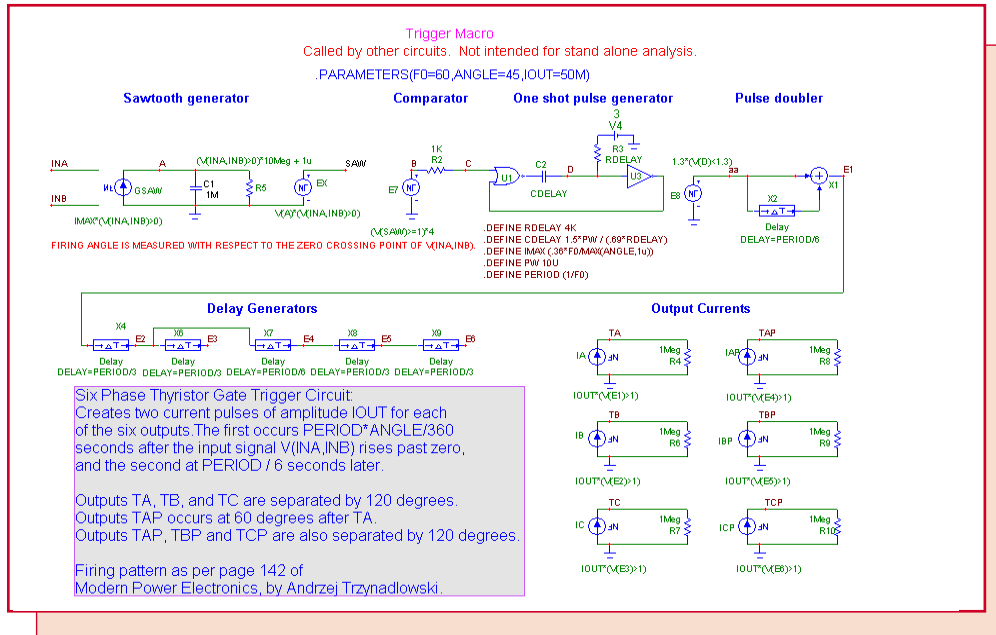
- The output voltage can be negative.
- Some firing angles can only be achieved with a suitable EMF voltage to assure SCR bias.
- For firing angles greater than 90 degrees and a negative EMF voltage the rectifier becomes an inverter and actually provides power to the AC sources.

To implement this type of rectifier/inverter from a simulation point of view we need:

- A set of three-phase AC sine sources
- A trigger generator that fires the SCRs
- An array of SCRS
- A suitable load

The first requirement is easily met by three SPICE V sources.

The second requirement is also easily met although with somewhat more work. Here is a macro circuit that does the job.



**Fig. 7- The trigger6 macro**

The sawtooth generator produces a voltage ramp that takes PERIOD\*ANGLE/360 seconds to charge capacitor C1 to 1.0 volts after the input differential voltage V(INA) - V(INB) crosses from a negative to a positive voltage. It does this because current GSAW is set to a suitable IMAX calculated as follows:

$$IMAX = C * DV/DT = 1m*1.0/(PERIOD*ANGLE/360)$$

$$IMAX = .360/(PERIOD*ANGLE) = .360*F0/ANGLE$$

where F0 is the sine wave frequency and ANGLE is the desired firing angle.

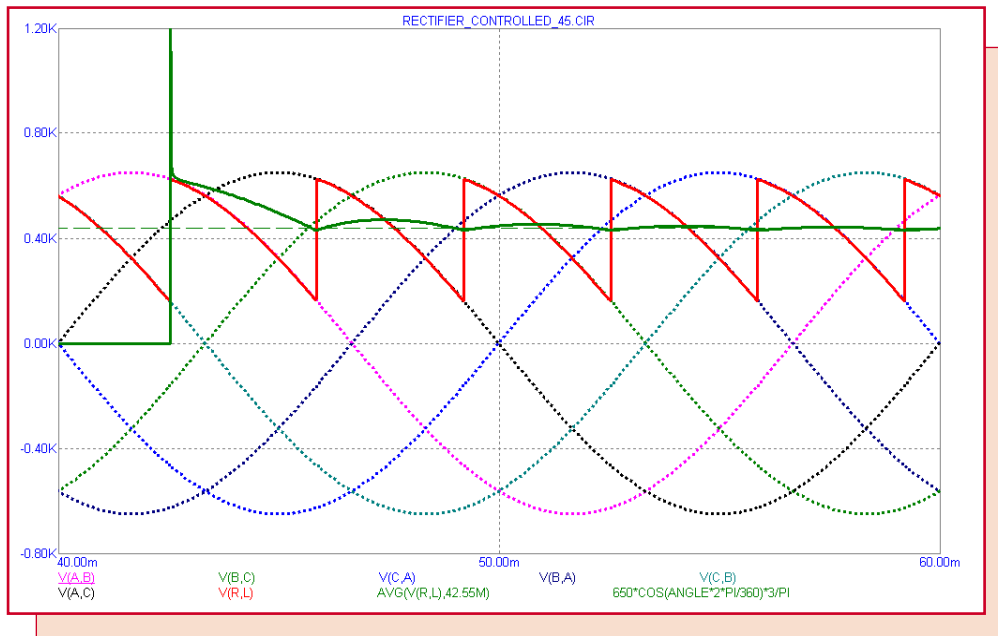
The comparator E7 detects when C1's voltage arrives at 1.0 volts and triggers a one-shot. The one-shot feeds a pulse doubler with produces two pulses with the necessary 60 degree separation. The pulse doubler then feeds a series of delays that create the remaining five trigger signals. Finally each trigger voltage pulse is converted to an output current pulse of magnitude IOUT.

The third requirement is an array of SCRs to control the load. These are arranged as shown and fired according to the pattern detailed on page 142 of reference 2.

- 1) XTA and XTBP fire to drive the load towards V(A,B)
- 2) XTA and XTCP fire to drive the load towards V(A,C)
- 3) XTB and XTCP fire to drive the load towards V(B,C)
- 4) XTB and XTAP fire to drive the load towards V(B,A)
- 5) XTC and XTAP fire to drive the load towards V(C,A)
- 6) XTC and XTBP fire to drive the load towards V(C,B)

The final requirement is a load, which usually consists of an inductance, a resistance and an EMF load such as a DC motor.

The waveforms for a firing angle of 45 degrees look like this:



**Fig. 8 - The transient analysis run for ANGLE=45**

Shown here are the three input signals and their inverses.

V(A,B) V(B,C) V(C,A) V(B,A) V(C,B) V(A,C)

The analysis is run from 0 to 60ms, but only the last 20 ms is shown to avoid the initial transients and show the steady state response.

In this circuit we have set up the following phase angles for the input sources:

VA: 30 degrees

VB: 30 - 120 = -90 degrees

VC: 30 - 2\*120 = -210 degrees

With this arrangement, the signal V(A,C) (black plot) is zero at T=0.

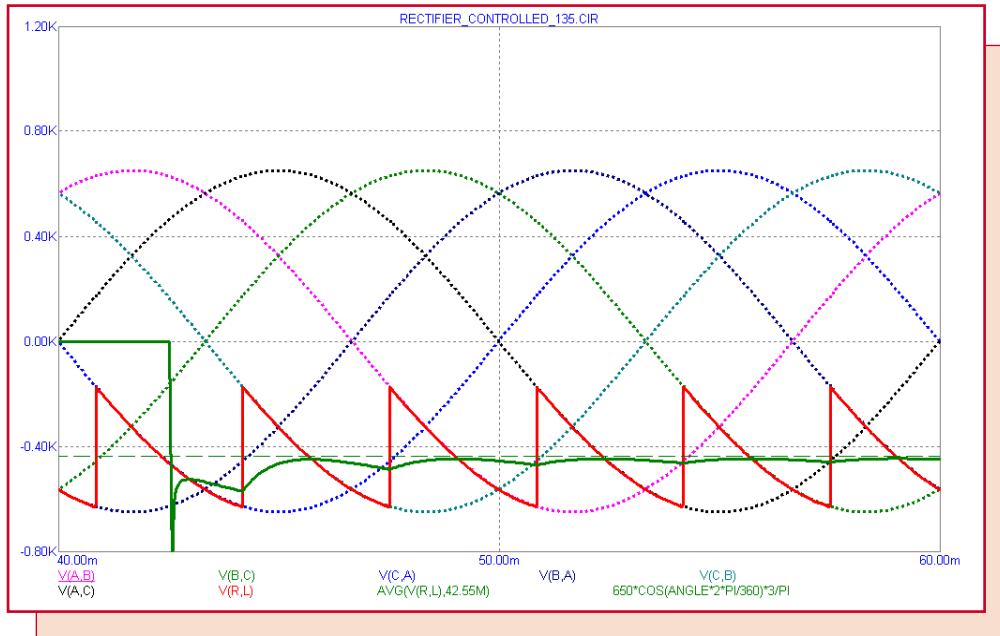
Superimposed on these is the output voltage across the load V(R,L) (red plot). Its average value (thick green plot) is also plotted and roughly matches the given formula (thin green line).

$$V_{OUT}(AVG) = (3/\pi)(V_{PEAK}) * \cos(ANGLE) - 2.0$$

In this version of the formula we have subtracted a 1.0 volt drop across each of the two SCRs.

The average function should be computed over many intervals or be started at the beginning of a charge cycle for its result to match the above formula.

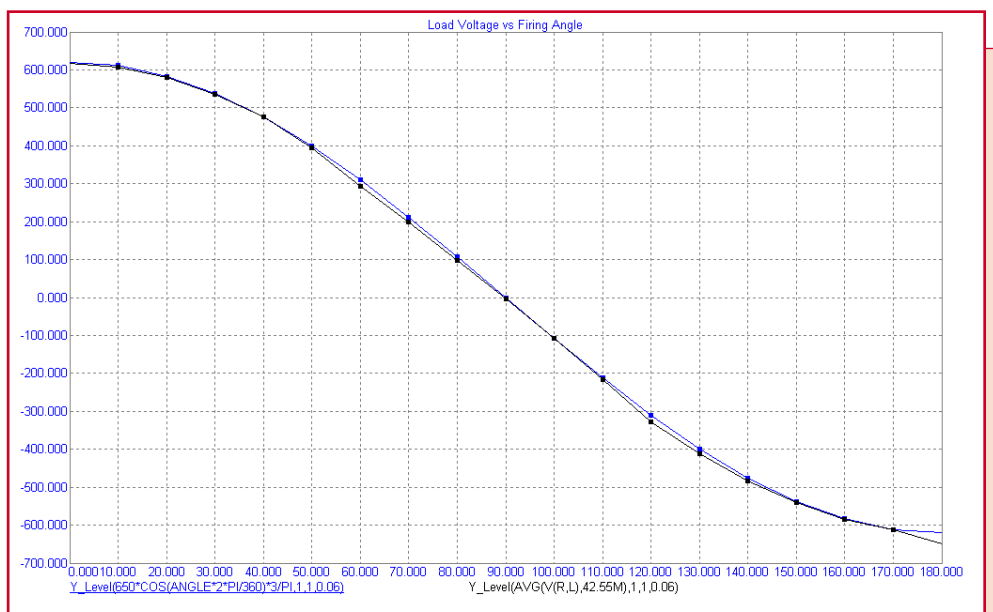
The next figure shows the waveforms for a firing angle of 135 degrees.



**Fig. 9- The transient analysis run for ANGLE=135**

Note that the output voltage is now negative. Since the current can only flow in one direction, the power flow is opposite of the 45 degree case. Power is now flowing from the EMF source to the 3 phase AC source.

The performance plot below shows the average output voltage across the RLE load at the end of the run for angles of 0 to 180 degrees. The plot shows both the theoretical formula and the actual average at the end of the run.



**Fig. 10- Theoretical and actual average load voltage vs. firing angle**

---

## Product Sheet

### Latest Version numbers

Micro-Cap 7 ..... Version 7.16  
Micro-Cap 6 ..... Version 6.33  
Micro-Cap V ..... Version 2.1.2

### Spectrum's numbers

Sales ..... (408) 738-4387  
Technical Support ..... (408) 738-4389  
FAX ..... (408) 738-4702  
Email sales ..... sales@spectrum-soft.com  
Email support ..... support@spectrum-soft.com  
Web Site ..... <http://www.spectrum-soft.com>  
User Group ..... micro-cap-subscribe@yahogroups.com