Conversores CC-CC com Isolamento Eléctrico

Objectivo

Utilizando o simulador PSpice 8.0 pretende-se realizar e verificar o funcionamento de dois conversores CC-CC com isolamento eléctrico: *Flyback* e *Forward.*

Conhecimentos Práticos Necessários

Este trabalho apoia-se na matéria leccionada na disciplina de Electrónica de Potência. A bibliografia recomendada é a dessa disciplina.

PSpice

É um *software* de simulação da MicroSim Corporation que ajuda a análise de circuitos eléctricos e electrónicos. Em anexo (anexo 2) é fornecida informação sobre esta ferramenta e respectivas instruções.

Introdução

Os conversores CC-CC são bastante utilizados em fontes de alimentação reguladas e em accionamentos de motores CC. Geralmente, a tensão de entrada destes conversores é obtida através da rectificação CA-CC da tensão da rede, sendo por isso flutuante devido às variações da tensão da rede de distribuição de energia eléctrica. Os conversores CC-CC são usados para converter essa tensão CC não regulada, numa tensão controlada.

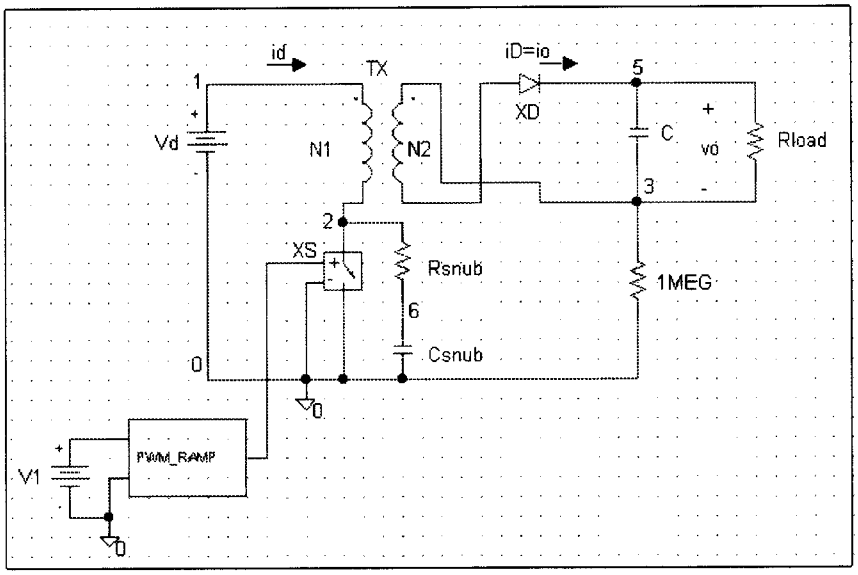
As fontes de tensão reguladas são necessárias para a maior parte dos sistemas electrónicos analógicos ou digitais. Pretende-se que a maior parte delas tenham os seguintes requisitos:

- Tensão de saída constante (dentro de uma tolerância pré-definida) para variações dentro da gama especificada da tensão de entrada e da carga.
- Isolamento eléctrico entre a entrada e a saída.
- Múltiplas saídas isoladas umas das outras.

Atendendo a estes requisitos, o principal objectivo é reduzir o tamanho da fonte de tensão e melhorar a sua eficiência. O avanço na tecnologia dos semicondutores permitiu-nos usar fontes de tensão comutadas, em oposição às fontes de tensão lineares. Como as fontes comutadas operam a frequências elevadas (componentes de filtragem menores), tornam-se menos pesadas e volumosas.

A comparação económica entre fontes lineares e comutadas depende da potência necessária.

Conversor CC-CC *Flyback*



Valores nominais:

$Vd = 32\text{V}$, $vo \approx 4\text{V}$, $D = 0.4$, $f_S = 200\text{kHz}$, $C = 100\mu\text{F}$, $R_{Load} = 1\Omega$.

Transformador: $N_1/N_2 = 4$, indutância de magnetização $L_m = 30\mu\text{H}$, indutâncias de fugas desprezáveis.

Exercícios:

1. Obtenha as formas de onda de v_I, v_O, i_d, e i_D.
2. Obtenha as formas de onda de v_I, i_{sw}, e i_D, durante a comutação.
3. Calcule os valor médios de i_d, e i_D e verifique que $\frac{I_d}{I_o} \approx \frac{V_o}{V_d}$ (justifique).
4. Observe a forma de onda de tensão no interruptor (v_{SW}) com e sem *snubber*.

   Verifique que o valor máximo da tensão é $v_{sw} = \frac{V_d}{1-d}$
5. Modifique os valores dos componentes de modo ao conversor funcionar no modo de condução descontínuo.
6. Verifique o ficheiro de saída de PSpice e procure interpretar os diferentes comandos nele contidos.

Notas:

1. O transformador é representado por um símbolo que se chama TRANS_NONIDEAL e que se encontra na biblioteca PWR_ELEC.SLB. Uma vez que as indutâncias de fugas são desprezáveis, o coeficiente de acoplamento k é aproximadamente igual a 1 (faça, p. ex., $k = 0.999$), pelo que:

   $$L_1 = L_m = 30\mu H \quad \text{e} \quad L_2 = \frac{L_1}{(N_1/N_2)^2} = 1.875\mu H$$

   Admita que o valor inicial da corrente em L_1 é 1.17A (faça ICOIL1 = 1.17A) e em L_2 é 0A.

2. Utilize como interruptor electrónico o componente representado pelo símbolo SWITCH da biblioteca PWR_ELEC.SLB.

3. Coloque um snubber R-C em paralelo com o interruptor. Faça:

   $R_{SNUB} = 100\Omega$, $C_{SNUB} = 1nF$

4. Devido a requisitos de conectividade do PSpice (não são permitidos nós flutuantes) deve ligar uma resistência de valor elevado (1MΩ, p. ex.) entre os nós 3 e 0.

5. Seleccione para díodo o subcircuito SW_DIODE_WITH_SNUB identificado pelo símbolo com o mesmo nome da biblioteca PWR_ELEC.SLB (ver anexo 1).

6. Para produzir o sinal de comando do interruptor utilize o subcircuito PWM_RAMP e introduza os seguintes valores para os parâmetros requeridos (ver anexo 1):

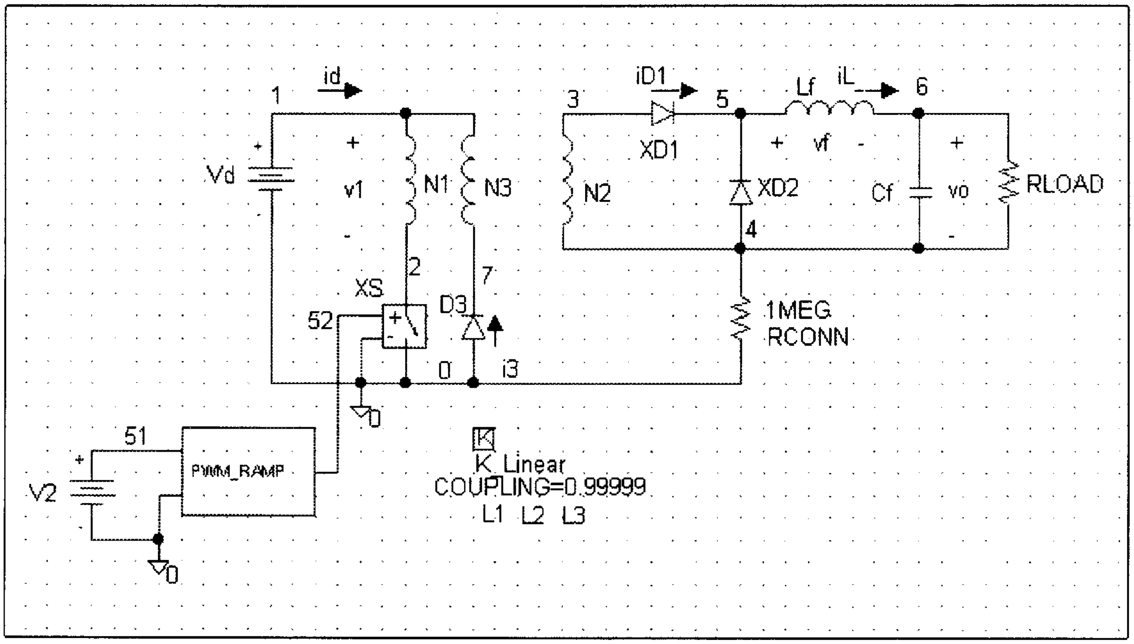
   RISE = 4.8µs, FALL = 0.1µs, PW = 0.1µs, PERIOD = 5µs.

   Para fazer variar o *duty cycle D* entre 0 e 1, a tensão de entrada de PWM_RAMP deve variar entre 0V e 1V.

7. Admita que o valor inicial da tensão no condensador de saída é 4.15V (faça IC = 4.15V).

8. Faça a análise do comportamento do circuito em regime transitório para, p. ex., 8 períodos de comutação com intervalos de tempo de 0.1µs.

Conversor CC-CC *Forward*



Valores nominais:

$Vd = 50V$, $vo \approx 4.5V$, $D = 0.4$, $f_s = 200kHz$, $L_f = 7.5\mu H$, $C = 100\mu F$, $R_{Load} = 1\Omega$.

Transformador: $N_1/N_2 = 4$, $N_1/N_3 = 1$, indutância de magnetização $L_m = 100\mu H$, indutâncias de fugas desprezáveis.

Exercícios:

1. Obtenha as formas de onda de i_L e da tensão de entrada do andar de saída (ou seja, a tensão aos terminas do díodo D_2.
2. Obtenha as formas de onda de v_1, i_{sw}, e i_3. Mostre que o valor médio de v_1 é igual a zero.
3. Mostre que $\frac{t_m}{T_S} = \frac{N_3}{N_1} D$, onde t_m é o intervalo de tempo durante o qual D_3 conduz.
4. Modifique (menu *Edit Model*) os parâmetros modelo do díodo D_3 (POWER_DIODE) para: $C_{JO} = 0.001fF$, $I_S = 1E\text{-}6A$, $R_S = 0.01\Omega$. Observe de novo a forma de onda de v_1. Que conclui?

Notas:

1. O transformador de 3 enrolamentos é representado por 3 indutâncias L_1, L_2 e L_3, com acoplamento magnético quase perfeito. Utilize o símbolo *K_Linear* da biblioteca de símbolos ANALOG.SLB para definir o acoplamento linear entre as 3 indutâncias. Uma vez que as indutâncias de fugas são desprezáveis, o coeficiente de acoplamento k é aproximadamente igual a 1 (faça, $k = 0.99999$), pelo que:

$$L_1 = L_m = 100\mu H, \quad L_2 = \frac{L_1}{(N_1/N_2)^2} = 6.25\mu H \text{ e } L_3 = \frac{L_1}{(N_1/N_3)^2} = 100\mu H$$

2. Devido a requisitos de conectividade do PSpice (não são permitidos nós flutuantes) deve ligar uma resistência de valor elevado (1MΩ, p. ex.) entre os nós 4 e 0.

3. Seleccione para díodos D_1 e D_2 o subcircuito SW_DIODE_WITH_SNUB identificado pelo símbolo com o mesmo nome da biblioteca PWR_ELEC.SLB (ver anexo 1).

4. Seleccione para díodo D_3 o subcircuito POWER_DIODE da mesma biblioteca.

5. Para produzir o sinal de comando do interruptor utilize o subcircuito PWM_RAMP e introduza os seguintes valores para os parâmetros requeridos (ver anexo 1):

   RISE = 4.8μs, FALL = 0.1μs, PW = 0.1μs, PERIOD = 5μs.

   Para fazer variar o *duty cycle D* entre 0 e 1, a tensão de entrada de PWM_RAMP deve variar entre 0V e 1V.

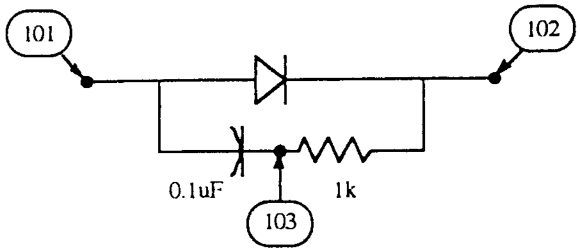
6. Considere que o valor inicial da corrente da indutância de saída (L_f) é 3A e que o valor inicial da tensão de saída é 4.5V.

7. Faça a análise do comportamento do circuito em regime transitório para um intervalo de tempo de 0 a 20μs, com intervalos de tempo de 0.1μs. Não permita intervalos de simulação superiores a 0.01μs.

ANEXO 1- Modelos dos Subcircuitos

SUBCKT DIODE_WITH_SNUB

Objective: This subcircuit is used to represent line-frequency diodes.

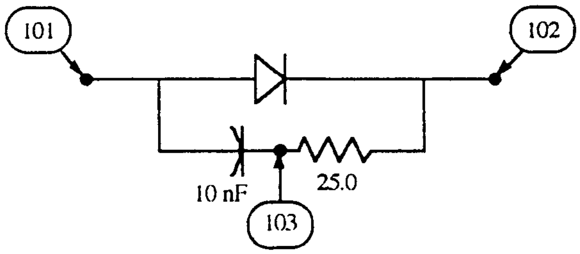


Input Circuit File Listing:

```
.SUBCKT      DIODE_WITH_SNUB    101     102
* Power Electronics: Simulation, Analysis & Education.....by N. Mohan.
DX           101     102     POWER_DIODE
RSNUB        102     103     1000.0
CSNUB        103     101     0.1uF
.MODEL       POWER_DIODE            D(RS=0.01, CJO=100pF)
.ENDS
```

SUBCKT SW_DIODE_WITH_SNUB

Objective: This subcircuit is used to represent sitching-frequency diodes.

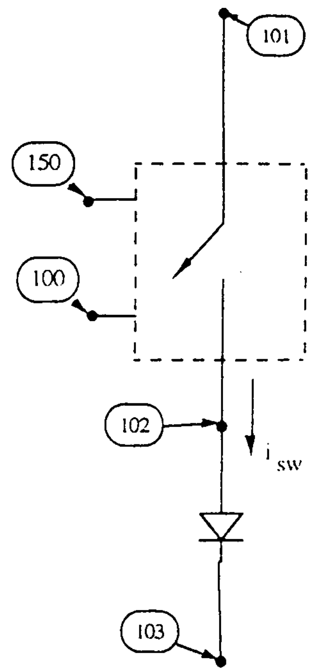


Input Circuit File Listing:

```
.SUBCKT      SW_DIODE_WITH_SNUB        101     102
* Power Electronics: Simulation, Analysis & Education.....by N. Mohan.
DX           101     102     POWER_DIODE
RSNUB        102     103     100.0
CSNUB        103     101     10nF
.MODEL       POWER_DIODE            D(CJO=0.001fF, IS=1E-6, RS=0.01)
.ENDS
```

SUBCKT SWITCH

Objective: This subcircuit is used to represent voltage-controlled switches through which the current is unididirectional.

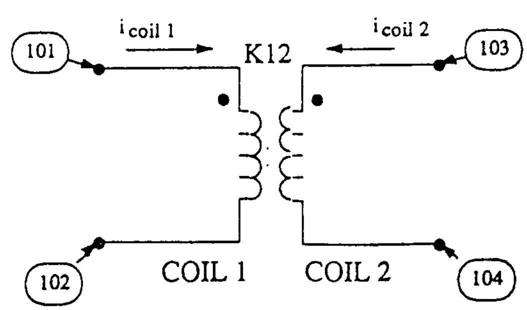


Input Circuit File Listing:

```
.SUBCKT     SWITCH       101    103    150    100
* Power Electronics: Simulation, Analysis & Education.....by N. Mohan.
SW          101    102   150    100    AC_SWITCH
DSW         102    103   POWER_DIODE
.MODEL      AC_SWITCH    VSWITCH ( RON=0.01 )
.MODEL      POWER_DIODE        D(CJO=0.001fF, IS=1E-6, RS=0.01)
.ENDS
```

SUBCKT TRANS_NONIDEAL

Objective: A nonideal tranformer.

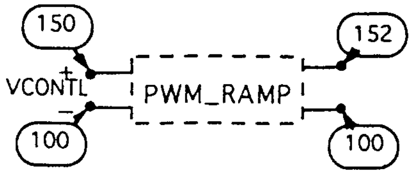


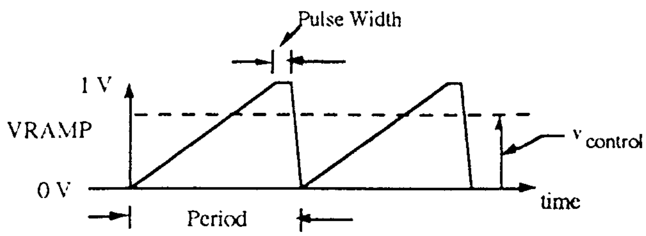
Input Circuit File Listing:

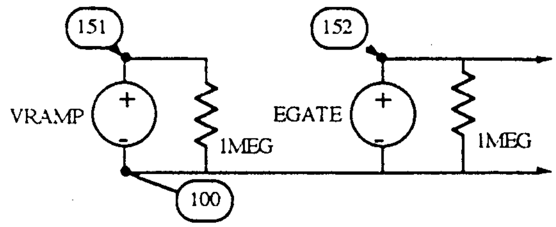
```
.SUBCKT     TRANS_NONIDEAL  101  102  103  104  PARAMS: ICOIL1=1A ICOIL2=1A
* Power Electronics: Simulation, Analysis & Education.....by N. Mohan.
LCOIL1      101    102    {L1}    IC={ICOIL1}
LCOIL2      103    104    {L2}    IC={ICOIL2}
K12         LCOIL1        LCOIL2        {k}
.ENDS
```

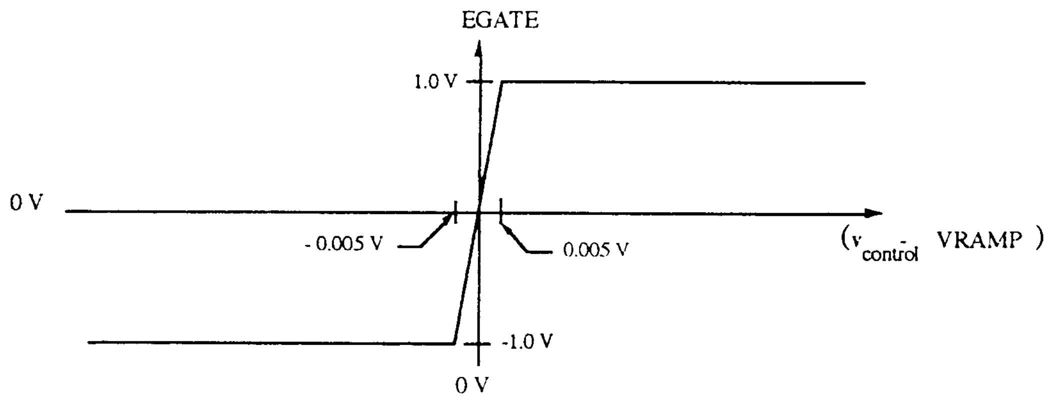
PWM_RAMP

Objective: This subcircuit is used for PWM control of single-switch dc-dc converters.









Input Circuit File Listing:

```
.SUBCKT     PWM_RAMP   150     100     152
* Power Electronics: Simulation, Analysis & Education.....by N. Mohan.
RCNTL       150     100     1MEG
RRAMP       151     100     1MEG
VRAMP       151     100     PULSE(0 1V 0 {RISE} {FALL} {PW} {PERIOD})
EGATE       152     100     TABLE { V(150) -  V(151) } = (-1.0,-1.0) (-0.005,-1.0) (0.0,0.0)
+                           (0.005,1.0) (1.0  ,          1.0)
RGATE       152     100     1MEG
.ENDS
```

Note: A finite pulse width PW is used to avoid voltage convergence problems.

ANEXO 2

Informação sobre o PSpice

Simulation Examples

2

Chapter Overview

The examples in this chapter provide an introduction to the methods and tools for entering circuit designs, running simulations with PSpice, and analyzing simulation results using Probe. All analyses are performed on the same example circuit. This allows for a clearer illustration of analysis setup, simulation, and result analysis procedures for each analysis type.

This chapter includes the following sections:

Example Circuit Design Entry

This section describes how to use MicroSim Schematics to enter the simple diode clipper circuit shown in Figure 2-1.

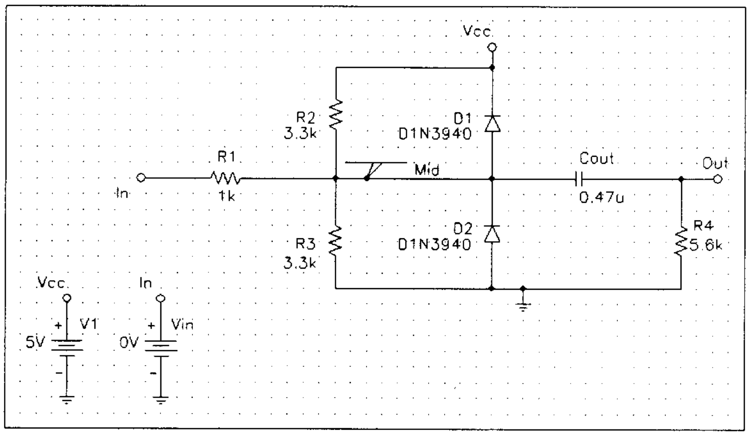
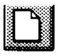


Figure 2-1 *Diode Clipper Circuit*

To open a new schematic window

1 Start Schematics. If Schematics is already running, be sure you are in the schematic editor.

2 If you are in a blank schematic window (indicated by "new" in the title bar at the top of the window), you can begin entering the circuit.

New icon: 

If you need to open a new schematic window, click the New icon or select New from the File menu.

To place the voltage sources

Shortcut: Click on  or press

1 Select Get New Part from the Draw menu to display the Part Browser dialog box.

2 Type VDC in the Part Name text box.

If you have enough room on your screen, click on Place to leave the Part Browser dialog box open.

3 Click on Place & Close.

4 Move the cursor to position the source at the desired location on the schematic page.

5 Click to place the first source.

6 Move the cursor and click again to place the second source.

7 Right-click to end placement mode.

To place the diodes

1 Go to the Part Browser dialog box.

2 Type D1N39* in the Part name text box.

3 Press Enter to display a list of diodes.

4 Click on D1N3940.

5 Click on Place (to leave the dialog box open) or Place & Close (to close the dialog box).

6 Press Ctrl+ r to rotate the diode outline to the desired orientation.

7 Click to place the first diode (D1), and click again to place the second diode (D2).

8 Right-click to end placement mode.

If needed, click on  to redisplay the Part Browser dialog box.

When placing components:

- Leave space to connect the components with wires.
- If device names and values don't match the names shown in Figure 2-1, they can be changed later.

To move the text associated with the diodes (or any other object)

1 Click once on the text to select it.

2 Drag the text to the desired location.

Select Redraw from the View menu or press Ctrl+ l to redraw the circuit as necessary.

To place the other components

Follow similar steps as described for the diodes to place the components listed below. The symbol names you need to type in the Part name text box of the Part Browser dialog box are shown in parentheses.

1 resistors (R)

2 capacitor (C)

3 ground symbols (EGND)

4 bubble symbols (BUBBLE)

To connect the components

Shortcut: Click on [icon] or press Ctrl+W.

1. Select Wire from the Draw menu to enter wiring mode. The cursor changes to a pencil.

You can right-click at any time to stop the wiring mode. The cursor changes to the default arrow.

If necessary, double-right click or press Spacebar to resume wiring mode. The cursor changes back to a pencil.

2. Click on the connection point (the very end) of the pin on the bubble at the input of the circuit.
3. Click on the nearest connection point of the input resistor R1.
4. Connect the other end of R1 to the output capacitor.
5. Connect the diodes to each other and to the wire between them:
   - **a** Click on the connection point of the anode for the lower diode.
   - **b** Move the cursor straight up and click on the wire between the diodes. The wire terminates and the junction of the wire segments is made visible.
   - **c** Click again on the junction to continue wiring
   - **d** Click on the end of the upper diode's cathode pin.
6. Continue connecting components until the circuit is wired as shown in Figure 2-1 on page 2-2.

Clicking on any valid connection point terminates a wire. A valid connection point is shown as an "x" (see Figure 2-2).

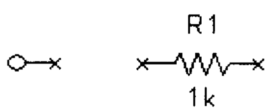


Figure 2-2 *Connection Points*

If you make a mistake when placing or connecting components:

1. Click on the wire or component.
2. Press Del.

To assign names (labels) to the nets and bubbles

1. Double-click on any segment of the wire which connects R1, R2, R3, the diodes, and the capacitor.
2. Type `Mid` in the Label text box.
3. Click on OK.

Bubbles serve as "wireless" connections where connectivity is implied by identical labels.

4. Double-click on each bubble to label it as shown in Figure 2-1 on page 2-2.

To place the viewpoint symbol

1 Type VIEWPOINT in the Part Name text box of the Part Browser dialog box.

2 Place the viewpoint so that its connection point (pin end) touches the wire labeled Mid.

To assign a name to a device

A particular name can be assigned to a device in the schematic (like "Vin" for a VDC, or Cout for the capacitor) as follows:

1 Double-click on the reference designator for the device.

2 Type the new name in the Edit Reference Designator dialog box.

3 Click on OK.

Example: assigning the name "Vin" for the second VDC.

1 Double-click on the reference designator of the VDC symbol, V2.

2 Type `Vin` in the Edit Reference Designator dialog box.

3 Click on OK.

To change the value of a device

1 Double-click on the value for the device.

2 Type the new value in the Edit Value dialog box.

3 Click on OK.

To save your schematic

1 Select Save from the File menu.

2 Type CLIPPER as the schematic file name.

3 Click on OK to save the file as `clipper.sch`.

Shortcut: Click on  or press [Ctrl]+ `s`.

Bias Point Analysis

Running PSpice

Shortcut: Click on  or press F11.

After creating the schematic `clipper.sch`, you can run PSpice by selecting Simulate from the Analysis menu. PSpice performs the simulation and generates the output file (`clipper.out`).

While PSpice is running, the progress of the simulation is displayed in the PSpice simulation status window (see Figure 2-3).

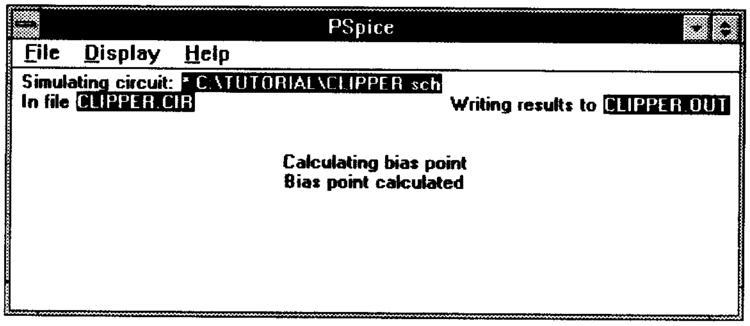


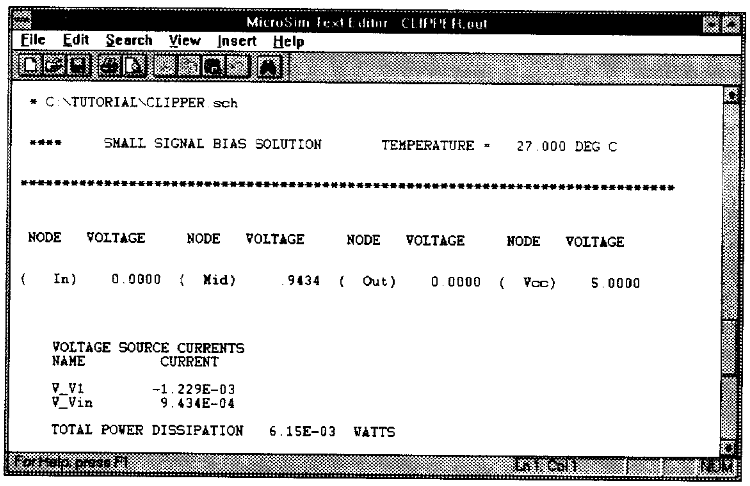
Figure 2-3 *PSpice Simulation Status Window*

Simulation Output File

The simulation output file acts as an audit trail of the simulation. This file optionally echoes the contents of the circuit file as well as the results of the bias point calculation. If there are any syntax errors in the netlist declarations or simulation directives or anomalies while performing the calculation, error and/or warning messages are written to the output file.

To view the simulation output file

1. Close the PSpice window.
2. In Schematics, select Examine Output from the Analysis menu to display the output file in the MicroSim text editor window. Figure 2-4 shows the results of the bias point calculation as written in the simulation output file (`clipper.out`).
3. When finished, close the MicroSim text editor window.




```
* C:\TUTORIAL\CLIPPER.sch

****     SMALL SIGNAL BIAS SOLUTION       TEMPERATURE =   27.000 DEG C

******************************************************************************

 NODE   VOLTAGE     NODE   VOLTAGE     NODE   VOLTAGE     NODE   VOLTAGE

(   In)    0.0000  (  Mid)      .9434  (  Out)    0.0000  (  Vcc)    5.0000


    VOLTAGE SOURCE CURRENTS
    NAME         CURRENT

    V_V1         -1.229E-03
    V_Vin         9.434E-04

    TOTAL POWER DISSIPATION   6.15E-03  WATTS
```

Figure 2-4 *Simulation Output File*

Since the diodes are both reverse biased (off), and the input source Vin is 0V (a short circuit to ground), the bias point is dependent only on the values of VCC, R1, R2, and R3.

The voltage at net Mid is in agreement with manual calculation:

$$V(MID) = \frac{Req}{R2 + Req} \times VCC$$

where:

$$Req = \frac{R1 \times R3}{R1 + Req}$$

Correct, expected bias point analysis results provide assurance of proper circuit connectivity.

Note that the current through VIN is negative. By convention, PSpice measures the current through a two terminal device into the first terminal and out of the second terminal. For voltage sources, current is measured from the positive terminal to the negative terminal; this is opposite to the positive current flow convention and results in a negative value.

Voltage Viewpoint

The VIEWPOINT symbol was initially a blank line. When the simulation is finished and bias point voltages are available, the viewpoint reflects the voltage at the net to which it was connected (in this case, the voltage at the node Mid is 0.9434).

Transient Analysis

This example shows how to run a transient analysis on the clipper circuit. This requires adding a time-domain voltage stimulus as shown in Figure 2-11.

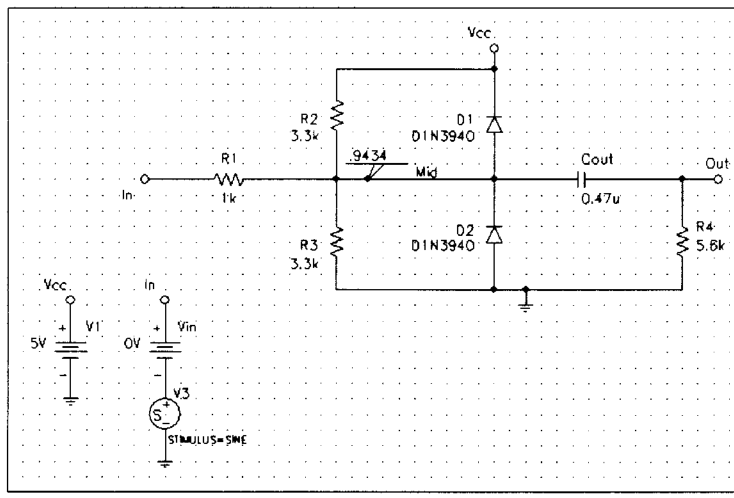


Figure 2-11 *Diode Clipper Circuit with a Voltage Stimulus*

To add a time-domain voltage stimulus

1 In Schematics, select Clear All from the Markers menu.

2 Select the ground symbol beneath the VIN source.

Shortcut: press Ctrl+x.

3 Select Cut from the Edit menu.

4 Scroll down or select Out from the View menu (or click on the Zoom Out icon).

5 Place a VSTIM symbol as shown in Figure 2-11.

6 Select Paste from the Edit menu.

7 Place the ground symbols as shown in Figure 2-10.

8 Select Fit from the View menu.

9 Select Save As from the File menu, and save the file as `clippert.sch`.

10 Double-click on the VSTIM symbol to start the Stimulus Editor.

If your version of Schematics does not include the Stimulus Editor (Basics users):

1 Place a VSIN symbol instead of VSTIM.
2 Double click on it.
3 Double-click on the appropriate attributes to set their values as VOFF, VAMPL, and FREQ. Click on Save Attr or press Enter after entering each attribute's value to accept the changes.
4 Click on OK.

11 When prompted to name the stimulus, type `SINE` and click on OK.

12 In the New Stimulus dialog box of the Stimulus Editor, click on SIN, and click on OK.

13 In the Stimulus Attributes dialog box, set the first three parameters as follows:

Offset Voltage = 0

Amplitude = 10

Frequency = 1kHz

14 Click on Apply to view the waveform. The Stimulus Editor window should look like Figure 2-12.

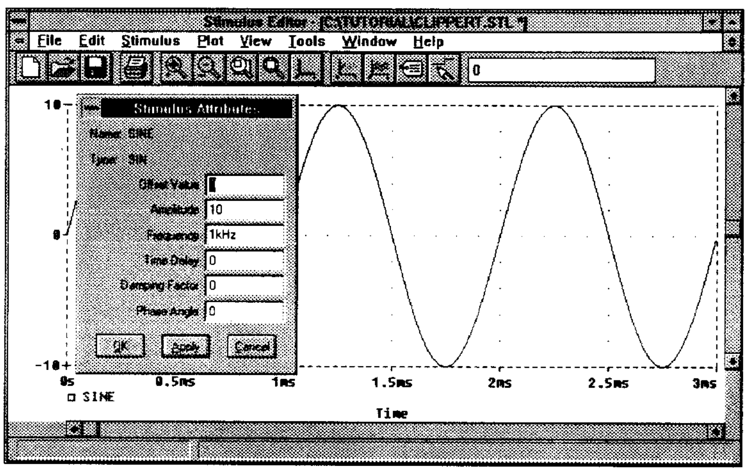
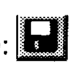


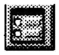
Figure 2-12 *Stimulus Editor Window*

15 Click on OK.

16 Click on the Save icon or press Shift+F12 to save the stimulus information.

Save icon: 

17 Select Exit from the File menu.

Analysis Setup icon: 

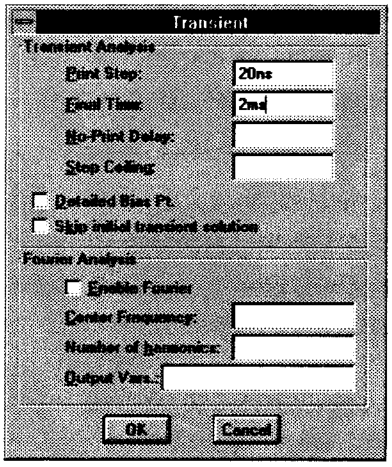
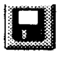


Figure 2-13 *Transient Analysis Dialog Box*

Save icon: 

Simulate icon: 

To set up and run the transient analysis

1. In Schematics, click on the Analysis Setup icon or select Setup from the Analysis menu to display the Analysis Setup dialog box.
2. Click on Transient to display the Transient Analysis dialog box.
3. Set up the Transient dialog box as shown in Figure 2-13.
4. Click on OK.
5. If needed, click on Transient check box in the Analysis Setup dialog box so that it is checked on (enabled).
6. If needed, disable DC sweep from the previous example by clicking on the DC Sweep check box so that it is *not* checked.

   DC Sweep is disabled here so that you can see the results of a transient analysis run by itself. PSpice can run multiple analyses during simulation, and you could run both DC sweep and transient analyses.
7. Click on Close to exit the Analysis Setup dialog box.
8. Click on the Save icon.
9. Click on the Simulate icon or press F11 to start the simulation.

PSpice uses its own internal time steps for computation. The internal time step is adjusted according to the requirements of the transient analysis as it proceeds. Data is saved to the Probe data file for each internal time step.

Note *The internal time step is different from the Print Step value. Print Step controls how often optional text format data is written to the simulation output file (.OUT).*

To display the output variable list with aliases and display desired traces in Probe

This portion of the example describes how to view the input sine wave and the clipped wave at Out. The output variable list is used as an alternative to markers to select the traces to be displayed.

1 Select Add from the Trace menu. (You can also click on the Add Trace icon or press Insert.)

2 Select V(Out) and V(In) by clicking on them in the trace list.

3 Click on OK. The traces are displayed.

4 Place the symbols shown in the trace legend on the traces themselves as shown in Figure 2-14:

   a Select Options from the Tools menu.

   b Click on Always in the Use Symbols portion of the dialog box.

   c Click on OK.

5 Click on the Save icon.

Save icon: 

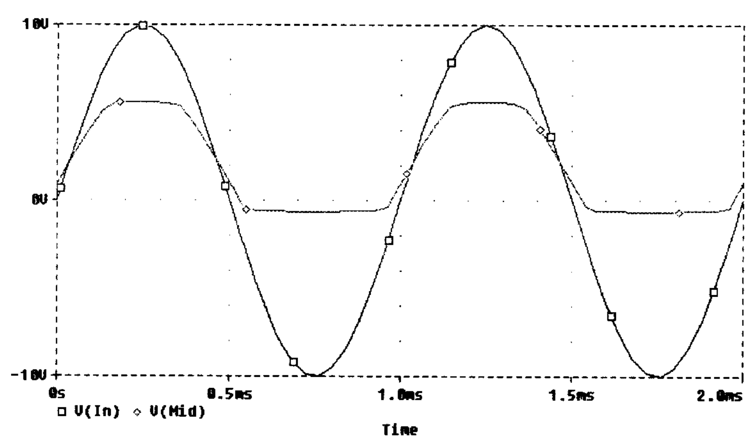


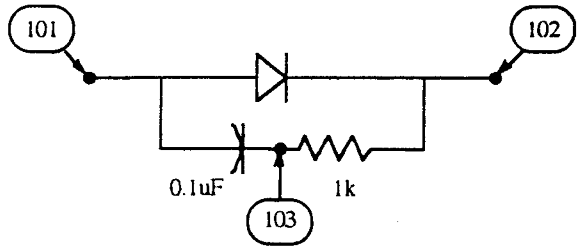
Figure 2-14 *Sinusoidal Input and Clipped Output Waveforms*

These waveforms illustrate the clipping of the input signal.

ANEXO - Modelos dos Subcircuitos

SUBCKT DIODE_WITH_SNUB

Objective: This subcircuit is used to represent line-frequency diodes.

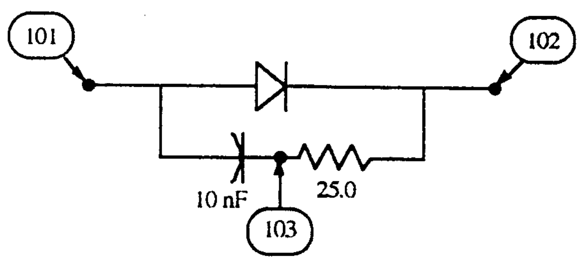


Input Circuit File Listing:

```
.SUBCKT    DIODE_WITH_SNUB    101    102
* Power Electronics: Simulation, Analysis & Education.....by N. Mohan.
DX         101    102    POWER_DIODE
RSNUB      102    103    1000.0
CSNUB      103    101    0.1uF
.MODEL     POWER_DIODE          D(RS=0.01, CJO=100pF)
.ENDS
```

SUBCKT SW_DIODE_WITH_SNUB

Objective: This subcircuit is used to represent sitching-frequency diodes.

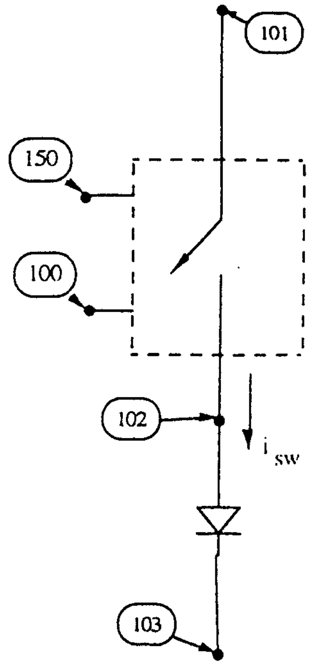


Input Circuit File Listing:

```
.SUBCKT    SW_DIODE_WITH_SNUB    101    102
* Power Electronics: Simulation, Analysis & Education.....by N. Mohan.
DX         101    102    POWER_DIODE
RSNUB      102    103    100.0
CSNUB      103    101    10nF
.MODEL     POWER_DIODE          D(CJO=0.001fF, IS=1E-6, RS=0.01)
.ENDS
```

SUBCKT SWITCH

Objective: This subcircuit is used to represent voltage-controlled switches through which the current is unididirectional.

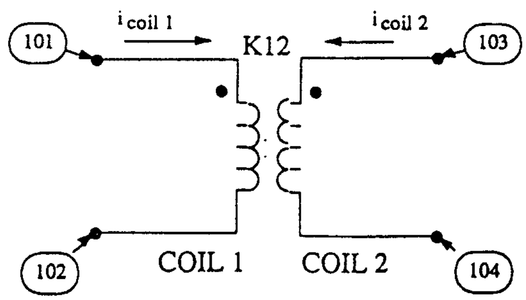


Input Circuit File Listing:

```
.SUBCKT    SWITCH       101    103    150    100
* Power Electronics: Simulation, Analysis & Education.....by N. Mohan.
SW         101    102   150    100    AC_SWITCH
DSW        102    103   POWER_DIODE
.MODEL     AC_SWITCH    VSWITCH ( RON=0.01 )
.MODEL     POWER_DIODE         D(CJO=0.001fF, IS=1E-6, RS=0.01)
.ENDS
```

SUBCKT TRANS_NONIDEAL

Objective: A nonideal tranformer.

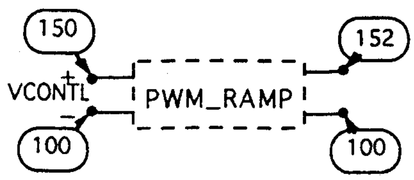


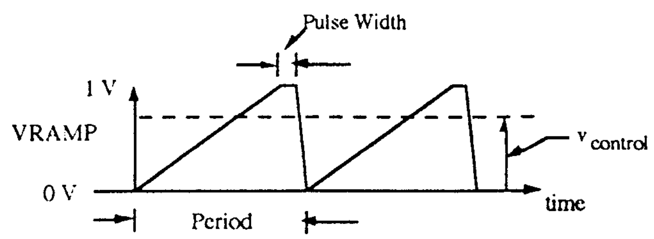
Input Circuit File Listing:

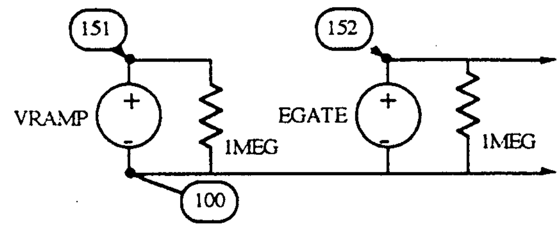
```
.SUBCKT    TRANS_NONIDEAL  101  102  103  104  PARAMS: ICOIL1=1A ICOIL2=1A
* Power Electronics: Simulation, Analysis & Education.....by N. Mohan.
LCOIL1     101    102    {L1}   IC= {ICOIL1}
LCOIL2     103    104    {L2}   IC= {ICOIL2}
K12        LCOIL1        LCOIL2        {k}
.ENDS
```

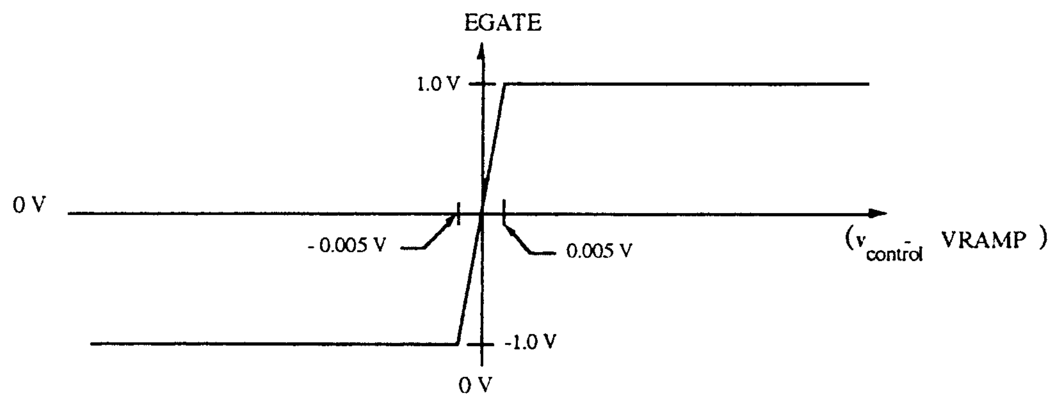
PWM_RAMP

Objective: This subcircuit is used for PWM control of single-switch dc-dc converters.









Input Circuit File Listing:

```
.SUBCKT     PWM_RAMP   150    100    152
* Power Electronics: Simulation, Analysis & Education.....by N. Mohan.
RCNTL       150    100    1MEG
RRAMP       151    100    1MEG
VRAMP       151    100    PULSE(0 1V 0 {RISE} {FALL} {PW} {PERIOD})
EGATE       152    100    TABLE { V(150) -  V(151) } = (-1.0,-1.0) (-0.005,-1.0) (0.0,0.0)
+                         (0.005,1.0) (1.0  ,       1.0)
RGATE       152    100    1MEG
.ENDS
```

Note: A finite pulse width PW is used to avoid voltage convergence problems.