International Journal of Engineering, Science and Mathematics
Vol.7Issue 4, April2018,
ISSN: 2320-0294 Impact Factor: 6.765
Journal Homepage: http://www.ijesm.co.in, Email: ijesmj@gmail.com
Double-Blind Peer Reviewed Refereed Open Access International Journal - Included in the International Serial Directories Indexed & Listed at: Ulrich's Periodicals Directory ©, U.S.A., Open J-Gage as well as in Cabell's Directories of Publishing Opportunities, U.S.A

THOUGHT-PROVOKING SIMULATIONS WITH THREE TERMINAL FIXED VOLTAGE REGULATORS

A.J. Chaudhari*
R.B. Waghulade**

Abstract

Generally, three terminal fixed voltage regulators are designed to provide fixed value of regulated voltages such as 5V,9V,12V etc. They are significant parts in most of the solid-state power supplies. In this paper we tried to simulate very thought-provoking applications other than regular ones. Mainly we emphasised on clipping and clamping applications using these regulators. For simulation and analyses of the circuits 5Spice software is used.

Keywords:

Fixed voltage regulator;
Clipping;
Clamping;
Simulation;
5Spice

Author correspondence:

A.J. Chaudhari,
Departmnet of Physics and Electronics,
M.J. College, Jalgaon-Maharashtra, India

1. Introduction

Three terminal fixed voltage regulators are commercially available in the marketthese voltage regulator ICs maintain the output voltage at a constant value. The LT1790-5 provides +5 volts regulated output voltage with provisions to add a heat sink.

SPICE (Simulation Program with Integrated Circuit Emphasis) is a general-purpose, analog and digital electronic circuit simulator programme. Large number of such softwaers are easily available either free, on trial or commercial. These includes LTspice, TINA, PSpice,5Spice, MULTISIM etc [1],[2]. In our study we used 5Spice.It is a Windows-based tool that supports a circuit designer to build circuits and simulations. Further perform analyses through a full graphical user interface (GUI). Fundamentally, it focusses on component level analog and digital circuit design and analysis [3]. Analysis in 5Spice consists of a Spice simulation such as: DC bias, DC, AC, Transient, Noise, Distortion.

*Associate Professor,M.J. College, Jalgaon-Maharashtra, India
**Principal, Shirish Madhukarrao Chaudhari College, Jalgaon-Maharashtra, India

2. Research Method

In this paper authors implemented simulation of fixed voltage regulators applicationsusing 5Spice software. It is general purpose simulation program with graphical interface.
Following main steps are employed for simulation:
1.Drawing the schematics
2.Setup Analysis-DC Bias, Transient
3.Run Analysis
4.Working with Graphs/Tables
5.Save all
After installation of software following window, shown in fig 1. will appear on the screen. It consists of three main areas. Schematic area is provided for drawing the circuit. Various components are available in the library. It consists of many categories. Toolbars are useful for selection, adding text conveniently etc [4],[5].

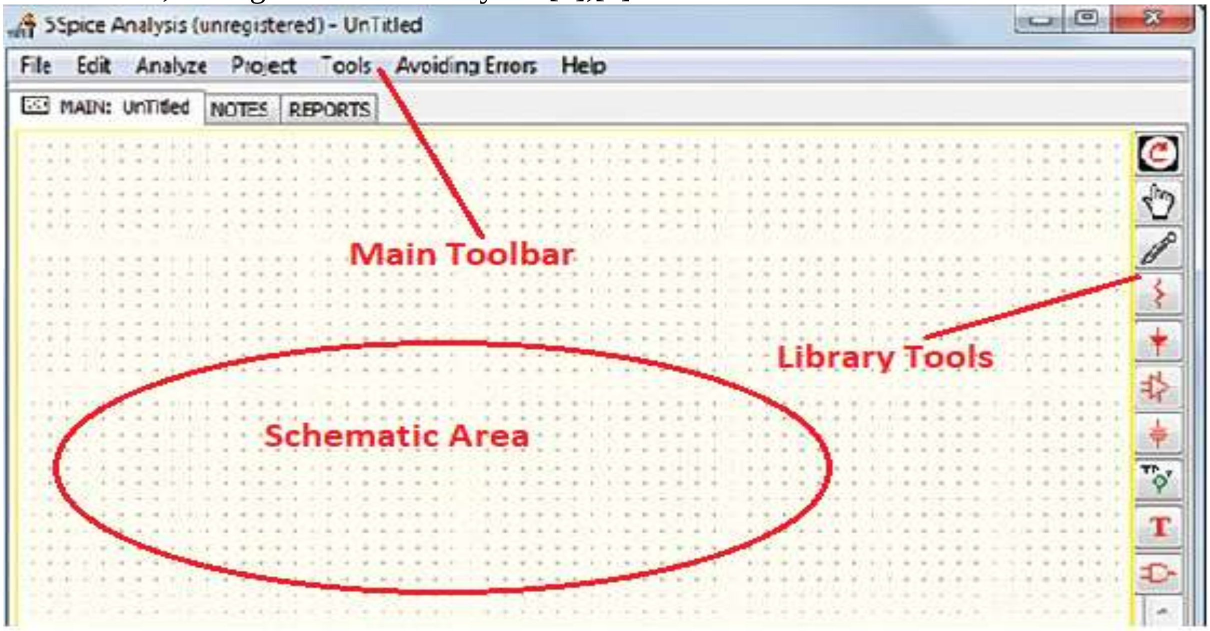


Figure 1. *Main Window of 5Spice*

5Spice Schematic for boosting the voltage:

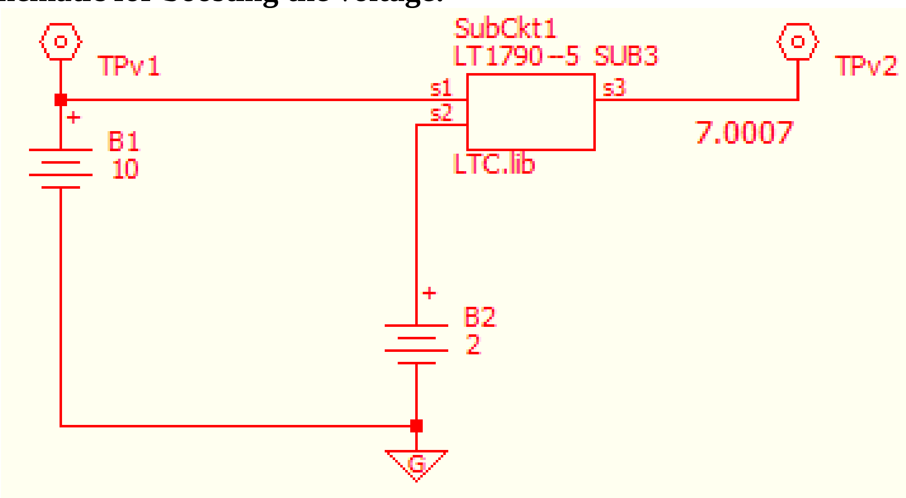


Figure 2. *LT1790-5 giving 7V*

Output of LT1790-5 gives constant output of 5V. Thus, if s2 is connected directly to ground we get 5V. But when s2 is connected to +2V as shown in Fig 2, addition takes place and DC Bias analysis results in to 7.0007 V regulated voltage.

Thus Vo(final) = Vo (fixed regulated) + 2V
= 5 + 2
= 7V

When battery voltage is reversed i.e. When s2 is connected to -2V as shown in fig 3,
Vo(final) = Vo (fixed regulated) + (-2) V
= 5 +(- 2)
= 3V

DC Bias analysis results in to 3.0013V.

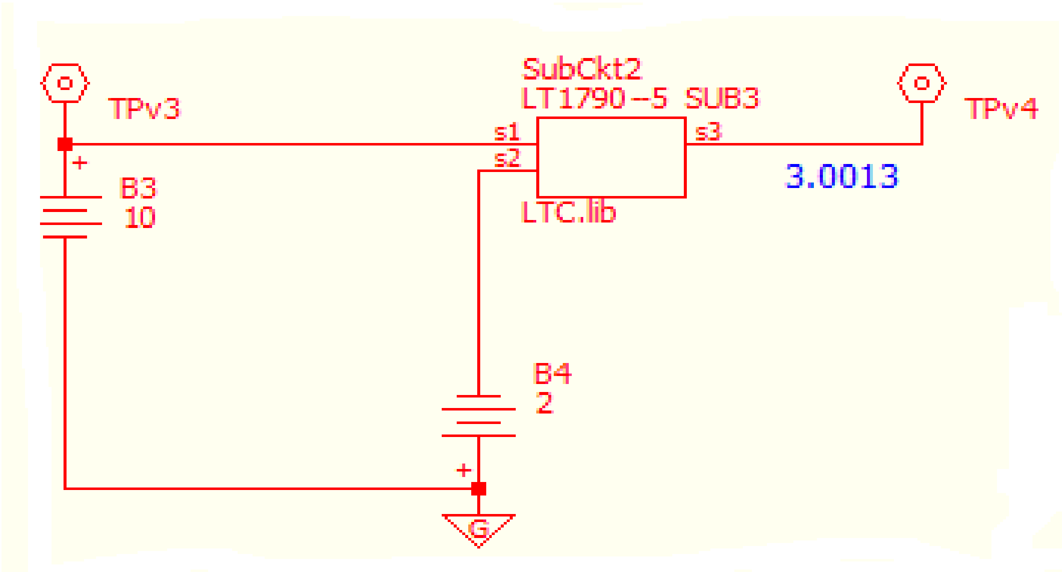


Figure 3. *LT1790-5 giving 3V*

Clamping the sinusoidal waveform:

Let us see the 5Spice sequence for exciting clamping application of LT1790-5.

=>START=>5Spice Analysis
=>File
=>New Schematic
=>Spice Library=>subcircuits,2-100pins
=>subcircuit1=>LT1790-5
=>signal source voltage Vs1=>TRAN/FFT analysis
=>sinewave
=>Amplitude 4V,
=> dc OFFSET 0V,
=>Frequency 500
=>constant voltage source, B1 12V
=>ground
=>wires=>connect
=>Test points TPv1, TPv2
=>Tools=>Generate Netlist

International Journal of Engineering, Science and Mathematics
Vol.7Issue 4, April2018,
ISSN: 2320-0294 Impact Factor: 6.765
Journal Homepage: http://www.ijesm.co.in, Email: ijesmj@gmail.com
Double-Blind Peer Reviewed Refereed Open Access International Journal - Included in the International Serial Directories Indexed & Listed at: Ulrich's Periodicals Directory ©, U.S.A., Open J-Gage as well as in Cabell's Directories of Publishing Opportunities, U.S.A

=> Avoiding errors=>schematic checklist
=>Analyze=>select analysis
=>Transient New
=>0 TO 2ms
=>Use source Vs1
=>Apply Changes
=>Apply and Run

The schematic for above application using 5Spice is as shown below in fig 4

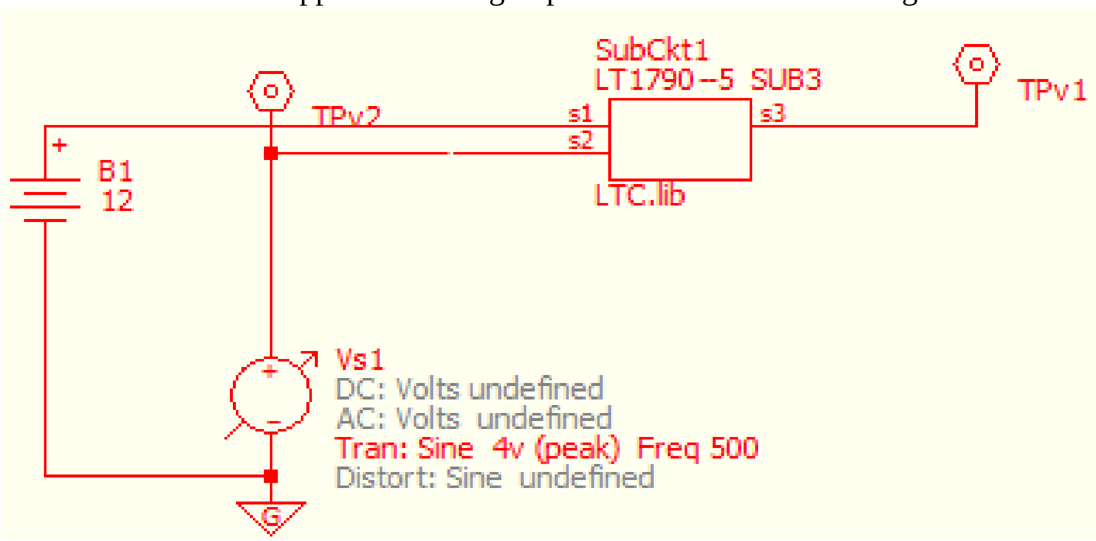
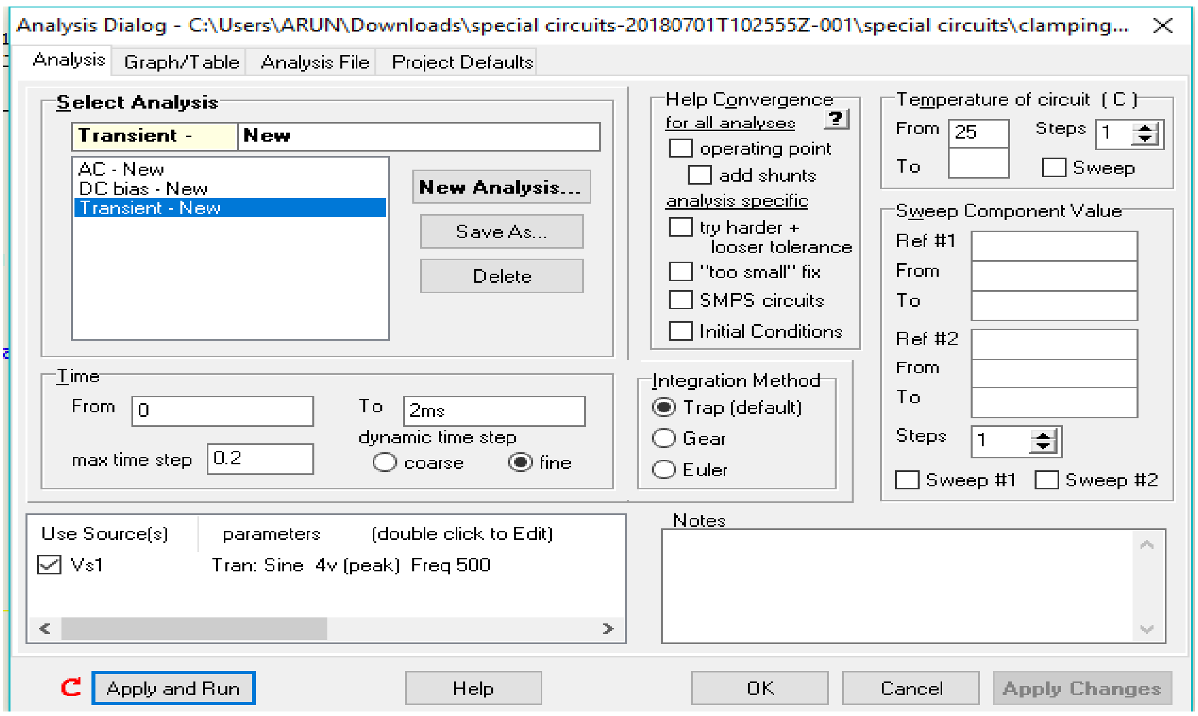


Figure 4. *5Spice schematic of LT1790-5 for clamping application*

The analysis setupis shown below in fig 5



International Journal of Engineering, Science and Mathematics
Vol.7Issue 4, April2018,
ISSN: 2320-0294 Impact Factor: 6.765
Journal Homepage: http://www.ijesm.co.in, Email: ijesmj@gmail.com
Double-Blind Peer Reviewed Refereed Open Access International Journal - Included in the International Serial Directories Indexed & Listed at: Ulrich's Periodicals Directory ©, U.S.A., Open J-Gage as well as in Cabell's Directories of Publishing Opportunities, U.S.A

Figure 5.*Analysis Dialog Window for clamping application*

Transient analysis [6] as shown in fig 6, clearly displays the clamping action. Wave is shifted above by 5V. Thus, wave is seen clamped at 5V.

Figure 6. *Clamping of sine wave*

Normally clamping and clipping circuits are constructed using diodes. Here we are simulating special circuits using LT1790-5 for these applications.
5Spice schematics of LT1790-5 for clipping action of sinusoidal waveform is shown below in Fig7.

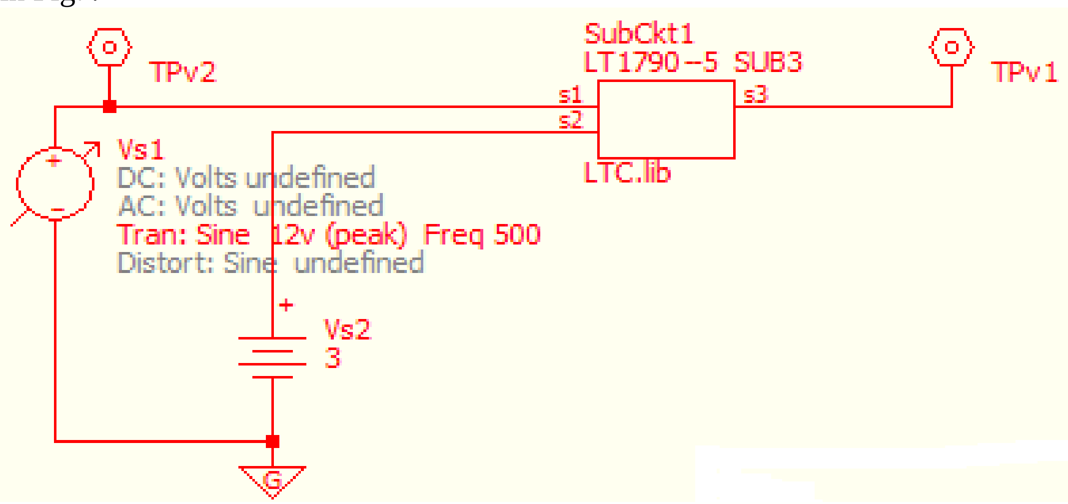


Figure 7. *5Spice schematic of LT1790-5 for clipping application*

The 5Spice sequence to generate this schematic is more or less similar as we discussed earlier in this paper. The complete sequence is given below.

Clipping the sinusoidal waveform:

Let us see the 5Spice sequence for exciting clipping application of LT1790-5.
=>START=>5Spice Analysis

=>File
=>New Schematic
=>Spice Library=>subcircuits,2-100pins
=>subcircuit1=>LT1790-5
=>signal source voltage Vs1=>TRAN/FFT analysis
=>sinewave
=>Amplitude 12V,
=> dc OFFSET 0V,
=>Frequency 500
=>constant voltage source, vs2 3V
=>ground
=>wires=>connect
=>Test points TPv1, TPv2
=>Tools=>Generate Netlist
=> Avoiding errors=>schematic checklist
=>Analyze=>select analysis
=>Transient New
=>0 TO 10ms
=>Use source Vs
=>sweep component value=>Ref #1 vs2
=>from 0 to 3
=> steps 2
=>ok
=>Apply Changes
=>Apply and Run

The analysis setup is shown below in fig 8.

International Journal of Engineering, Science and Mathematics
Vol.7Issue 4, April2018,
ISSN: 2320-0294 Impact Factor: 6.765
Journal Homepage: http://www.ijesm.co.in, Email: ijesmj@gmail.com
Double-Blind Peer Reviewed Refereed Open Access International Journal - Included in the International Serial Directories Indexed & Listed at: Ulrich's Periodicals Directory ©, U.S.A., Open J-Gage as well as in Cabell's Directories of Publishing Opportunities, U.S.A

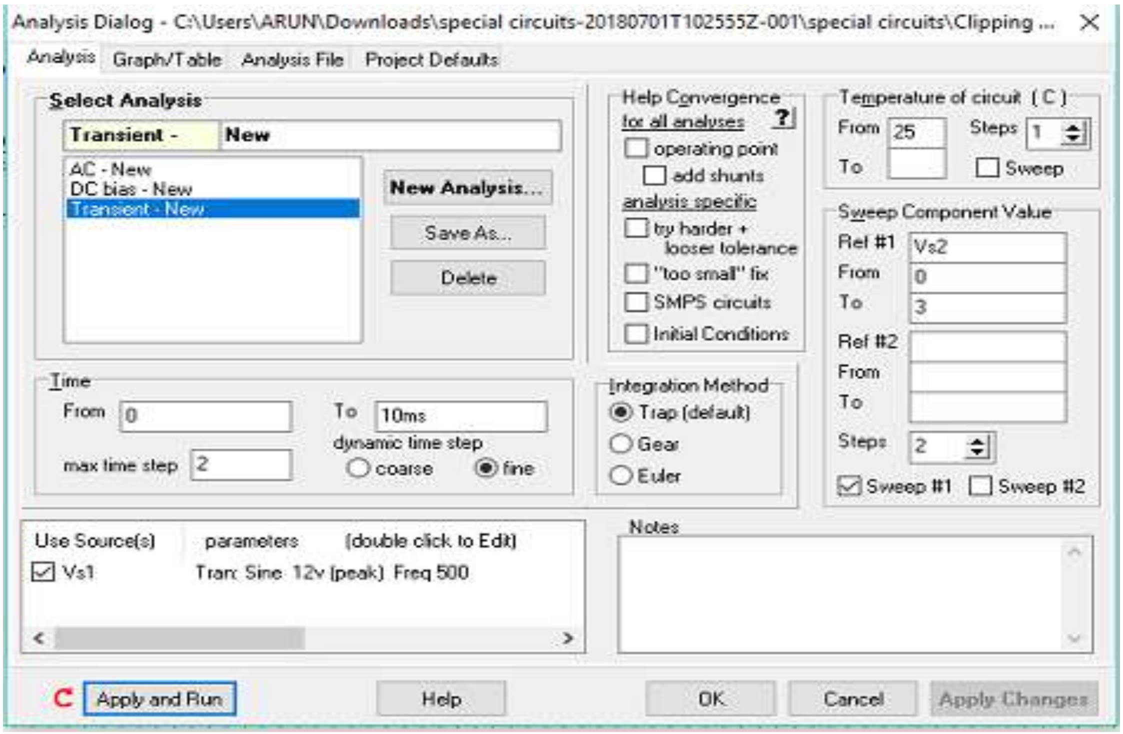


Figure 8. *Analysis Dialog Window for clipping application*

In this case sweep component value option is used to generate three different clipping levels. The transient analysis results in to various clipping levels such as 5V,6.5V and 8V for 12V input sinusoidal. It is shown in fig 9.

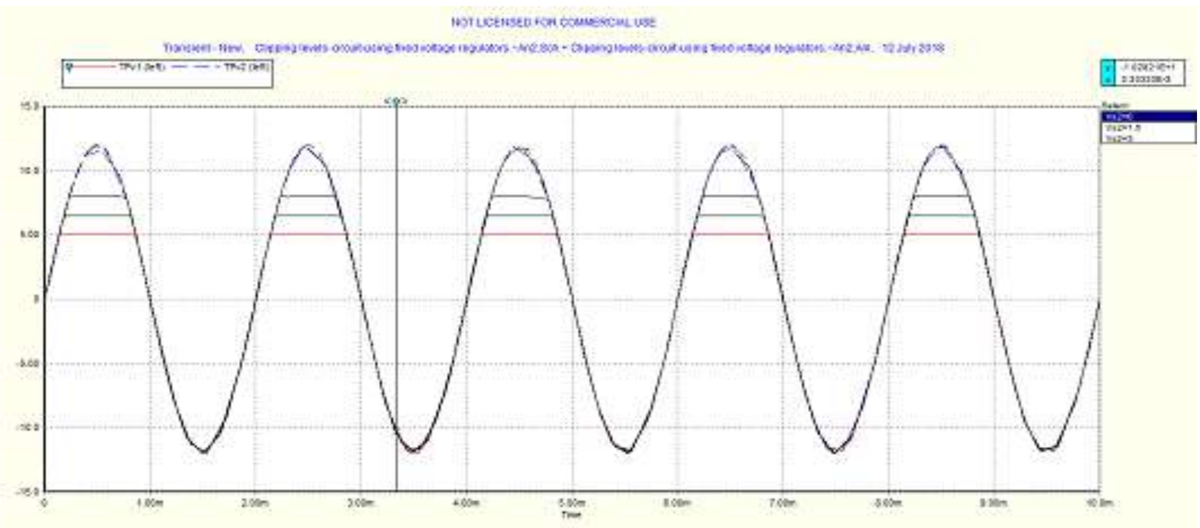


Figure 9. *Clipping the sine wave at different levels*

3.Result and Discussion:

Normally three terminal fixed voltage regulators are essentially used in designing power supplies. In our study we tried few new and innovative applications by using 5Spice simulator. DC Bias analysis is used to increase or decrease the fixed output of LT1790-05.
It appearances that the terminal s2 is of significantly important. When it is at ground level, 5V appears at output. But if it is at some other voltage then one can boost or decrease the output from its fixed value. Generally, diodes are used for applications such as clipping and clamping. Here we inventively use simulations and transient analyses for these applications.Due to 5V fixed voltage regulator sinusoidal waveform clamped at 5V. While different clipping levels 5V,6.5V,8V are obtained due to sweep of vs2 from 0 to 3V in 2 steps.

4.Conclusion:

Though three terminal fixed voltage regulators are designed to get regulated output volatages,5Spice simulator is successfully used for its clamping and clipping applications. DC Bias analysis supports to get outcomes in boosting the normal fixed output while transient analysis is desirable for clipping and clamping applications.

References

[1] https://electronicsforu.com/resources.
[2] https://www.edn.com/electronics-blogs/out-of-this-world-design/4438701/It-s-time-to-SPICE-up-your-life-
[3] http://www.ecircuitcenter.com/index.htm
[4] Muhammad H.R., Introduction to PSpice Using OrCAD,3rd Edition, pp.393-424, 2010
[5] Altium Designer-Module 21, Circuit Simulation
[6] 5Spice Manual