World Academy of Science, Engineering and Technology
International Journal of Energy and Power Engineering
Vol:12, No:11, 2018

Power Supply Feedback Regulation Loop Design Using Cadence PSpice Tool: Determining Converter Stability by Simulation

Authors : Debabrata Das

Abstract : This paper explains how to design a regulation loop for a power supply circuit. It also discusses the need of a regulation loop and the improvement of a circuit with regulation loop. A sample design is used to demonstrate how to use PSpice to design feedback loop to control output voltage of a power supply and how to check if the power supply is stable or oscillatory. A sample design is made using a specific Integrated Circuit (IC) available in the PSpice library. A designer can experiment feedback loop design using Cadence Pspice tool. PSpice is easy to use, reliable, and convenient. To test a feedback loop, generally, engineers use trial and error method with the hardware which takes a lot of time and manpower. Moreover, it is expensive because component and Printed Circuit Board (PCB) may go bad. PSpice can be used by designers to test their loop designs without using hardware circuits. A designer can save time, cost, manpower and simulate his/her power supply circuit accurately before making a real hardware using this software package.

Keywords : power electronics, feedback loop, regulation, stability, pole, zero, oscillation
Conference Title : ICPEPE 2018 : International Conference on Power Electronics and Power Engineering
Conference Location : Venice, Italy
Conference Dates : November 14-15, 2018