

Pcb design

Design



Is a convention to PC, subsequent construction and testing. The design of BPCS Is very widely used in electronic equipment and systems, for example, good EMCEE (Electromagnetic Compatibility) practices. These PC-level EMCEE practices usually help achieve the required EMCEE performance at a much lower cost than alternative EMCEE measures at higher levels of integration, such as whole-product shielding. They also improve signal integrity In both analogue and digital circuits (Armstrong, Gauge 1999) Ronald 1).

Power decoupling techniques can Improve the power supply of a PC and remission lines techniques are a part of the PC layout (Armstrong, Cot 1 999) Journal 2). The MUMMER is a miniature development module which uses Fad's FETTERS. In this assignment, MUMMER needed to be designed according to the dataset. Besides, switches of four pins also needed to be designed, but only two pins were used in diagram. PICKOFF 320, a microelectronic, which features six enhanced power-managed " software controlled" modes, power consumption as low as 0. 1 microcosms in standby mode and a wide operating voltage range from 2 to 5. Lots which makes this device Ideal for battery managed applications (Unknown, It has some advantages, such as namely and high computational performance at an economical price, with the addition of high endurance Enhanced Flash program memory. There are a range of features that can significantly reduce power consumption during operation, including Alternate Run Modes, Multiple Idle Modes, On-the-fly Mode Switching and Lower Consumption In Key Modules (Unknown, 2004) [3]. A pre-programmed PICNIC 8F1320 was

made available for testing purpose. This assignment is a simple two LED display, 3 push button switches and

USB interface PICNIC 8F1320 based PC. The Picket connector enables this device to be programmed with the user functions required. The Leeds keep flashing when the PC was connected with the power supply or USB cable. Pushing the reset button (WI), both Leeds go out for a moment then both come on for a moment and then start flashing again. The aim of this assignment is to make the flashing decrease (Increase) in frequency by pushing and holding down WWW (WWW). This assignment consists of three parts: PC design, PC construction and PC testing. 1. 2) Background 1. 3) Block Diagram

The basic building blocks of a boost converter circuit are shown in Fig 1. 1 . 1 . To begin with, MUMMER and three switches of four pins are needed to design in ARES, and then the MUMMER is needed to draw in SIS'S. Next packaging the MUMMER and three switches in ISIS. After the circuit diagram is completed, it should be PC layout. During the PC layout, all connectors should be near the edge of the PC for easy access. The PC will be powered from the MUMMER USB interface but when the MUMMER is not inserted power connection can be made to Jell and J Versions so make ere these two pins are far enough apart to avoid shorting power connections.

There should be room at each of the four corners of the PC for a rubber foot, which will require a mm hole. The rubber foot itself is mm in diameter and must be allowed for on the PC layout. 2. The Leeds should stand mm away from the PC surface The following shows the test schedule: 1 . Simulating the

circuit diagram by using the oscilloscope; 2. Testing PC without EPIC or MUMMER inserted; 3. Testing PC after EPIC was inserted and add the correct power connections; 4. Testing PC after MUMMER was also inserted and plugging in a USB cable to a EPIC laptop.) Possible solutions anywhere on the power rail. At the same time, check that the correct pins on the EPIC have OVA and +IV values, and check that the MUMMER connector has the correct voltage power values. 4) Implementation 4. 1) PC Design First of all, the MUMMER was designed according to its dataset in ARES and then it was packaged so that it can be found in ISIS. Next three switches (four pins) were designed and packaged by using the same methods, however, only two pins were used in the circuit diagram.

It needs to convert the schematic into a single side PC layout in ARES after the circuit diagram was completed. The PC was required ex. inches in size and the pads used for the DILL components were oval rather than the smaller circle that is the default in the ARES library in order to provide enough copper pad when soldering. All connectors were placed near the edge of the PC for easy access. The LEEDS stood mm away from the PC surface. In addition, Jell and J Versions were placed far enough apart to avoid shorting power connections.

Finally, Auber feet (mm hole) were required in the four corners of the PC, the rubber foot itself is mm in diameter. The final PC design sees Figure 4. 1 . The Leeds started flashing when the correct COM port number was set to 19200 Baud, with Flow control was set to none and a text message appeared on the PuTTY screen after pressed the reset button. (Implementation) 4. 2) PC Construction.

<https://assignbuster.com/pcb-design/>