

Creating Professional Circuit Designs Using NI Circuit Design Suite

Circuit design remains the main challenge to get an optimal performance for your controller board for any application. Here is how to approach your designs with professional tools and processes

■ ARUN DAYAL UDAI

One of the greatest things about professional robots or other electronic gadgets that fascinates us is their customised circuit boards with no hanging wire or protruding part. Such designs are reliable and rugged and can outperform any controller board that is not designed for any particular application or made up on breadboards.

With the advent of PCB prototyping machine and its availability, manufacturing such a board has become quite inexpensive. Moreover, with widespread use of mobile phones, technical knowhow for fixing SMD components is easily available. You can get the chips placed with precision from any mobile repair shop. Thus circuit design remains the only challenge where you need to concentrate to get an optimal performance for your controller board for any application. Here we show you how to approach your designs with professional tools and processes.

NI Circuit Design Suite is a well known tool that electronics engineering students have been using for long. It is mostly used for circuit simulation, through National Instrument's Multisim package, which is a module in NI Circuit Design Suite. NI Ultiboard is the next module to Multisim in NI Circuit Design Suite. Using Ultiboard you can physically design a circuit board layout, design shape of the base board, place different components and do routing of different electrical networks. A 30-day trial package can be downloaded for free from the website of National Instruments. Engineer-

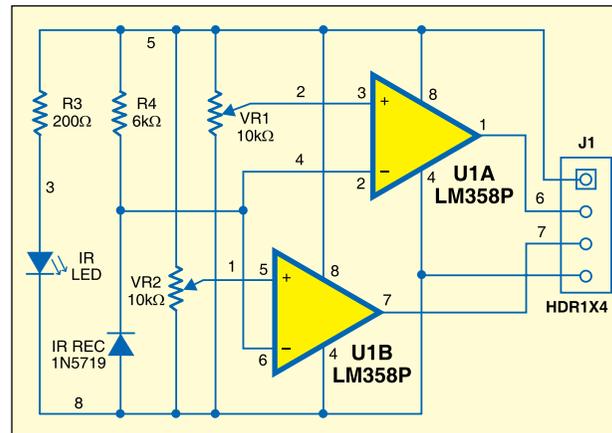


Fig. 1: Circuit for a line sensor

ing colleges may have its full version installed in their circuit simulation or design labs.

Typically, a circuit can be very easily constructed in Multisim environment and simulated to analyse different run-time parameters. Assuming that you know the circuit for any given application, you need to place the components on the board and route different networks. The best way I have always preferred to teach any subject is to start with an example and then practice with different sets. We will use Ultiboard, which is meant for the aforesaid purpose, with a commonly used IR line-sensing module. The circuit may look as shown in Fig. 1.

LM358 consists of two voltage comparators, U1A and U1B. Both of them are fed with two different reference voltages through potentiometers VR1 and VR2. The IR signal transmitted by the IR LED is sensed by a single IR-sensing module, and compared with two different threshold voltages through the two comparators. The out-

put through wires 6 and 7 is made available at the header. The header also supplies voltages for comparators and IR modules.

Normally, a line-sensing module utilises only one of the comparators for white-line detection. You may use both of them to work at different lighting conditions, without

putting much effort or additional components.

Designing a board is more of mechanical in nature than electronics. You need to know the footprints of all the devices you are going to use. This may require measuring the pin geometry using a Vernier calliper. Multisim and Ultiboard have very good library of various electronic devices and accessories, which includes their footprints.

The first step in circuit designing is to create a network of devices placed using wires, which is known as a 'netlist.' Two different approaches are followed for building the netlist for a given circuit, which is the primary requirement for designing a circuit. A simple approach is to create schematic on the Multisim workspace, by simply inserting the different components required for a circuit and connecting the leads through wires. This may be transferred to Ultiboard by pressing 'Transfer' tab in the main menu of Multisim. This creates a netlist file (*.ewnet). It will ask for default track

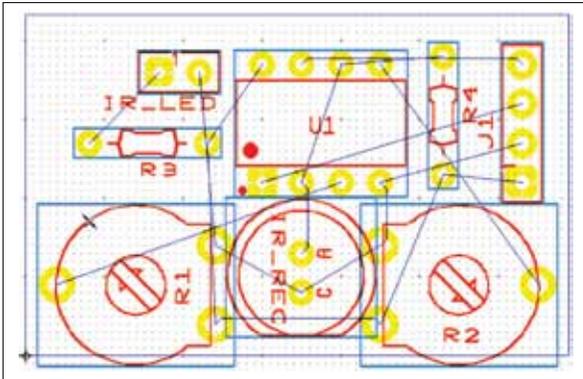


Fig. 2: Netlist created for the given circuit

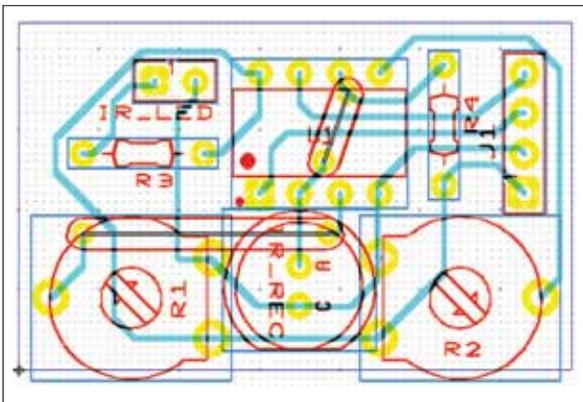


Fig. 3: Circuit board after routing of tracks

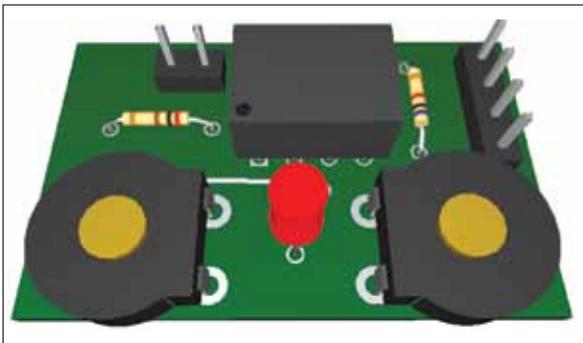


Fig. 4: 3-D view of the board

width and clearance to trace, which can be altered later also. Select units as millimetre for easy understanding and estimation at this stage. Track width should not be less than 0.5 mm if it is going to be manufactured by etching techniques. This generates a netlist file with a default double-sided board with all the components placed at its top edge. Once you reach this stage, you are ready to start board designing.

This can be achieved by an alter-

native technique that starts from Ultiboard directly. Create a single- or double-sided board of the required size by selecting 'Board Wizard' main menu tools. Now place different parts from the database. Here, more concentration is given to the shape of the parts rather than the actual name of the part. Now create a different netlist using netlist editor from 'Tools' menu. Select the pins that are in a particular net and the board layer (copper top or bottom) where it will lie. You can also adjust widths of the track formed by any particular net.

With the netlist formed, you can do auto-placement of all the devices by clicking Autoplace from Autoroute menu. This places the components within the board. However, this hardly helps. You are required to place and orient the components on the board taking care of various constraints, like a serial port socket or a power supply socket should lie on the edge of the board.

Similarly, a crystal should be placed as close as possible to a microcontroller. Voltage regulators should be fitted on the edge so that these can be connected to the heat-sink if required. Some components like headers or female headers require to be placed for easy handling. You must optimise the placements for maximum efficiency of the system.

Resize the board after placing of all the components is finalised. After creating the network and adjusting

the board size, the layout may look as shown in Fig. 2. If the footprint for any part is not available in the database, the same can be created using component wizard from 'Tools' menu, by customising various dimensions like pin width, gap, drill diameter and solder pad dimensions. These dimensions may be found in the datasheet provided with the device or can be measured. The solder pad sizes should be kept to 0.5mm annular space if the components are to be soldered manually.

After creating the netlist and configuring various board parameters like track width and clearances, you may go for routing of the tracks. This can be done by selecting Autorouter from 'Autoroute' menu. Some tracks may not get auto-routed. These can be completed manually by placing tracks or fixing jumpers at some locations. The tracks are generated automatically depending on certain optimisation techniques for least track length, least number of through-hole connectors (Vias), etc. The tracks further require manual adjustments like straightening of tracks and placing of silk-screens on top and bottom. The final board looks as shown in Fig. 3. The same can be visualised in virtual by selecting View3D from 'Tools' menu.

Now that you have completed everything in soft, exchange this design with the PCB prototyping machines. One of the most widely used formats for this data exchange is GERBER RS-274X. Our design from NI Circuit Design Suite can be exported to this format by selecting 'Export' in 'Files' menu. Properties of Gerber files should be adjusted to print each layer that exists in the current design. Most important layers are copper top, copper bottom, board outline, drill and silk-screens. These files can be sent to different manufacturers for PCB prototyping.

The basic processes for other designs remain similar. Only the complexity of the design may increase. ●

The author is a lecturer in Department of Mechanical Engineering, Birla Institute of Technology, Mesra, Ranchi