Developing Printed Circuits Board using Protel 99SE

Most designs today are built on Printed Circuit Boards (PCBs), which consist of multiple layers of electrical copper and insulating material sandwiched together. Several Electronic Design Applications (EDA) exist to both create schematics of a circuit and transfer them to a working PCB layout. At the University of Florida, we have access to Protel 99SE for these functions. The resulting PCB design can then be sent to a company for a professional board with silkscreens, multiple layers, etc. MIL also has the ability, through a T-Tech Quick Circuit, to make single and double sided circuit boards PCBs out of copper-plated sheets of insulating material. The latter process is unable to make complicated designs, but it both quick and cheap. This tutorial covers the creation of a sample design using Protel 99SE, and its preparation to be milled out by the T-Tech.

First we must create an overall database that will contain all the individual components of the design – the Schematics that describe the circuit, the PCB Designs that physically layout the components and their connections, and other assorted files.

- Start Protel and select **File/New**. Use a *MS Access Database* and give it a name – this will create a Design Database file (extension .DDB) that contains all the parts of your design. In this case, we'll call it *Tutorial.DDB*. Maximize the window – it makes it easier to have one title bar.

- The Database has its own internal File System – go into the Documents Folder, select **File/New** and create a new *Schematic Document*. Always rename your files to something more descriptive (*HC11 IO.sch*, *Power Regulation.sch*, etc.) - this will make it easier to keep track of what information is in each document. We'll stick to the pattern and call it *Tutorial.sch*.

- Save often – both Windows and Protel can be buggy at times, and the last thing you want to do is lose over an hours work because of a crash.

We can now create the schematic that will describe our circuit. This consists of placing symbols that represent th e individual components in the design and connecting their individual connections, or nodes, to one another.

- In the middle is the design window where we actually create our systems. You'll notice in the left panel three windows. The upper window shows what Schematic Libraries you currently have open, and the middle window shows individual components in the library selected from that list. If instead you see a list of files in your design database, then click the *BrowseSch* Tab, and set the pulldown menu to *Libraries*. Usually, the only open Library will be Miscellaneous. Devices. We will start by using some of its components to make a sample design.

- Select *CON4* from the middle window by double clicking it, and then move the pointer to the design window – you will see the outline of the component. Pressing the **SPACE BAR** will rotate the symbol. Pressing **TAB** will allow you modify the Properties of the Device. In this case, we go to the Properties, Select the **Graphical Attributes Tab**, and Click *Mirrored* - this will reverse the symbol about the y-axis. Click once to place the component on the left side of the sheet, about half way down.

- The Final Schematic is shown on Page 4 if any questions arise.

- Select **Place/Wire**. Move the Pointer on the end point of pin 1 on the connector. A Black Circle should appear, indicating that a connection between the wire and a node on the component can be placed. Click to start the wire. Move the cursor about 6 squares, and then click again. This will lock the wire segment and allow you to keep extending the wire in other directions. In this case, we press **ESC** to end the wire. Do the same for pins 2, 3 and 4.

- Select **Place/Net Label** and move back to the Design Window. A Box Outline should appear. Press **TAB** to go to properties, and enter IN1 for a *Net Name*, and close. Move the box right above the first wire so that the black circle shows up and click. Move down one row and click on the second node. It should automatically increment and place IN2. Do the same for the other two nodes. Net Labels will allow you to connect separate nodes on the page without actually laying a wire between them.

- Now Place a *CON3* in the upper left corner (mirror it as well). Connect Wire Segments to Pins 1 and 2. We will use this as a power connector. Select **Place/Power Port**, and go to properties. Select a *Net* of VCC, and use the *Bar Style*. Connect it to Pin 1. Place another Power Port – use a *Power Ground Style*, and call it GND. Connect it to pin 2. Power Ports are essentially Net Names with special symbols – they also will connect separate nodes on the page. Use **Place/Place Directive/No ERC** on pin 3 – this tells Protel nothing should be connected to that pin.

- We use a 3 Pin Connector for power even though there are only two connections. This will ensure that even if we accidentally plug in the connector backwards, that we will not destroy the board in the process. Keeping in mind that your creating a physical object and thinking ahead on how it will actually be used will save a lot of time when actually working with the PCB.

- Also place a *CON1* – connect a wire and net name to it called ENABLE.

- We will now place AND Gates in the design. To do this we need to use another library. Click **Add/Remove** under the Library Window. The Protel Schematic Libraries are in *C:\Program Files\Design Explorer 99SE\Library\Sch*. Double-Click on *NSC Databooks.DDB* and then click OK.

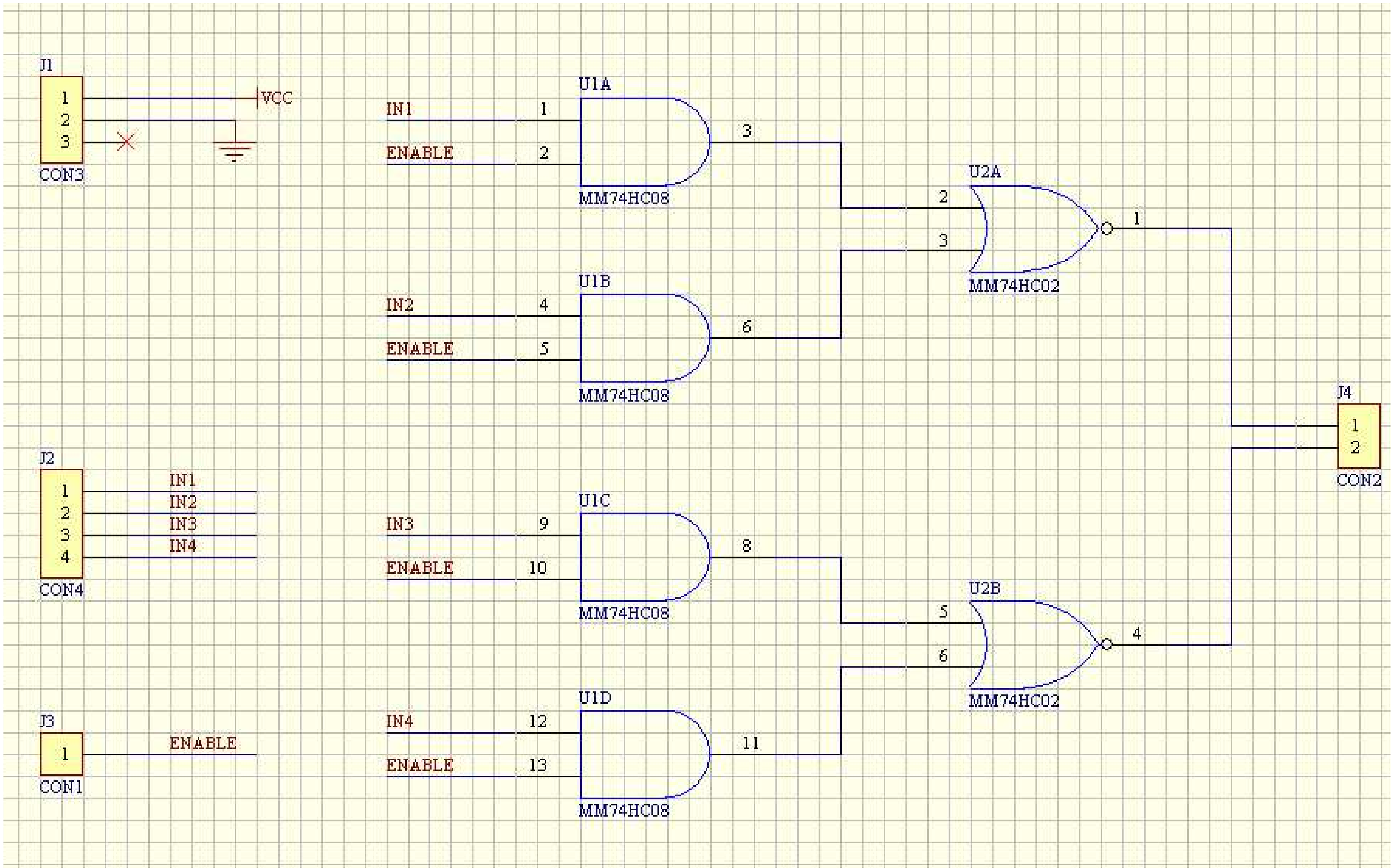
- Several more Libraries are now available. Select *CMOS Logic Databook 1988*, and then double click *MM74HC08* – an AND Gate. Using the filter will allow you to find it quicker – enter in the Filter Window **08**. Place one of the devices into the design, and extend wire segments from the nodes. Then click and drag until the gate and wires are surrounded by the outlined box, and let go. The symbols will be selected and therefore be outlined in yellow, and can be copied. You can select a single component by holding shift and then clicking it. **Merely clicking a symbol and having a dashed outline will let you delete it, but not let you copy it to the clipboard.** Press **CTRL-C** to copy and click the mouse on the start of one of the wire segments – when you paste the symbols, they will be oriented from this location. The symbols can be deselected by holding **X** and then pressing **A**.

- Paste the Gate three times so that you have four gates total. Connect the first inputs to net names of IN1 – 4, and the second inputs to ENABLE. Place two more gates – *MM74HC02*s, which are NOR Gates– to the left of the HC08s. Connect them to the AND Gates as shown on Page .

- If you look at the logic gates, you will notice that the AND gates are identical, as are the NOR gates – we are using a single chip for each device, which is obviously not a good idea. To use a single chip for multiple gates, we must go back to the properties window of each device and change the *Parts* from the default of 1 to 2, 3, and 4 as necessary – 4 gates are in the HC08, and both NOR gates can fit in a HC02. You will notice that the gate designators change from U?A to U?B, U?C, etc.

- We will now number these designators. This can be done using **Tools/Annotate**, which will automatically assign numbers to the designators. There are a few options in what parts to number – in this case, use the defaults. You will notice that all the AND Gates are under U1, and the NOR gates are U2.

- The Resulting Schematic is



Now that the schematic is complete, we must make sure there are no errors in the design. Then, we must prepare the design to be imported into the section of Protel that actually creates PCBs. Protel has multiple tools to do both.

- We are now ready to check our design to make sure there are no errors. This can be done using whats called an Electrical Rule Check (ERC). It can be found under **Tools/ERC**. There are many thinks the ERC can look for, though for this case, we are mainly concerned about accidental short circuits and the like. Therefore, use the defaults. The ERC should come back reporting no errors.

- We are now ready to actually create a PCB from this schematic. **There is no information about physical data in a schematic – orientation, spacing, and the like mean nothing.** The only data that will be sent to the PCB document besides a list of what nodes are connected to each other (a netlist) are the number of pins on each part, and the Footprint for each part. The latter is where physical data comes into play. A Footprint is a graphical symbol that shows where each pin of a device is physically located. We still must enter this information before we can create the PCB.

- The PCB Libraries are in *C:\Program Files\Design Explorer 99SE\Library\PCB.* While PCBs manufactured elsewhere create circuit boards that are covered with a laminating material, the T-Tech Machine at MIL creates circuit boards that are just exposed layers of copper. Therefore, soldering to these boards can be more difficult then soldering to a regular PCB. We therefore use special PCB libraries for T-Tech Boards – they are located in *C:\Program Files\Design Explorer 99SE\Library\Pcb\Generic Footprints\MLL_PCB.DDB.*

- Open up *MIL_PCB.DDB* – you'll see several .LIB files in the Documents folder. These are PCB Footprint Libraries. If you open up *SHEET1.LIB* and *SHEET1OB.LIB*, you will notice they have the same footprints, with the only difference being that the latter has oval, or oblong, pads (the dark gray) instead of circles like SHEET1. The Oblong pads are easier to solder to, and should always be used when possible if you are making a copper clad board.

- We use the following footprints for each part:

Part	Footprint	Library
CON1	SIP1R	SHEET1OB.LIB
CON2	SIP2R	SHEET1OB.LIB
CON3	SIP3R	SHEET1OB.LIB
CON4	SIP4R	SHEET1OB.LIB
MM74HC02	DIP14R	SHEET1OB.LIB
MM74HC08	DIP14R	SHEET1OB.LIB

- The Connectors Property Windows can be opened, and the Footprints entered (SIP1R, SIP2R, etc.) Instead of entering the Footprint for each logic gate, we can use the Global Feature. Open up the Properties Window of one of the HC08s, and click **Global**. The Window expands. The Middle Column lets us set attributes to select certain components, and the Last Column allows us to set certain properties of all those components that are selected. In this case, we determine that all parts that have a Library Reference of *MM74HC08* should have a DIP14R Footprint. There are entered in their boxes under the *Attributes to Match By* and *Copy Attributes* Columns, respectively. Once OK is clicked, it will tell you how many gates it found that should be modified, and ask if you want to go through with the operation. In this case, there should be four objects – if so, go ahead. All four AND gates should now be using DIP14R Footprints.

- Do the same for the two NOR Gates.

We are now done with creating our Tutorial Schematic. Up next is actually creating the PCB. We start by transferring the schematic information – a netlist and footprints – to a new PCB document. There footprints can be located in the available system libraries, or you can create your own PCB Library and make your own footprints for special parts.

- We start by going to **Design/Update PCB** in our schematic. A new PCB document is created (*TUTORIAL.PCB*), and a window pops up. Leave the defaults, except that you should unselect *Generate Component Class for all Schematic Sheets in Projec***t**. Then click **Preview Changes**.

- A Window should pop up showing all the modifications that will be made to the PCB Document. Click *Only Show Errors*. The Table should then be blank. If there are errors, its probably because Protel couldn't find a Footprint. Check your Schematic to make sure they're all entered correctly.

- Once there are no errors, click **Execute**. Then Select **View/Fit Documen**t**.** Your footprints should appear, with faint lines showing the connections between the components.

Before we can place the components on the PCB document, we must understand how the PCB is actually laid out, i.e. In layers.

- A PCB exists as Simple Graphical Constructs (Lines, Circles) on multiple layers. These Layers include
    - Top, Bottom Layer: Where Electrical Routes occur
    - Top, Bottom Overlay: Graphical Symbols Showing the Part Layout
    - Keepout Layer: Circuit Board Outline
    - Multi layer – Exists on all of the Layers

  There are other layers as well,, but we are most concerned with these when making boards at MIL. The T-Tech cannot do Overlay Layers, but it can do Double Sided Boards, meaning we can use both Top and Bottom Layers.

- The PCB Window shows the design in the center of the screen. Towards the left is an area similar the Browse Schematic Windows you previously saw. Here new footprint libraries can be added, individual parts and traces can be highlighted, or errors can be found. There is also a magnification window that lets you quickly move around the board, and a pull down window that sets the current layer you work in.

- We will normally create single-sided boards with the T-Tech, unless a design is so complex that it requires routes on both side. Therefore, all parts should be moved from the top layer to the bottom. This can be done easily using the global feature. Double-Click a footprint to get to its properties, and change the Layer to *Bottom Layer*. Click the **Global Tab**, and then hit OK. All the Components should change from a bright yellow outline to a dark tan.

We can now actually design the PCB.

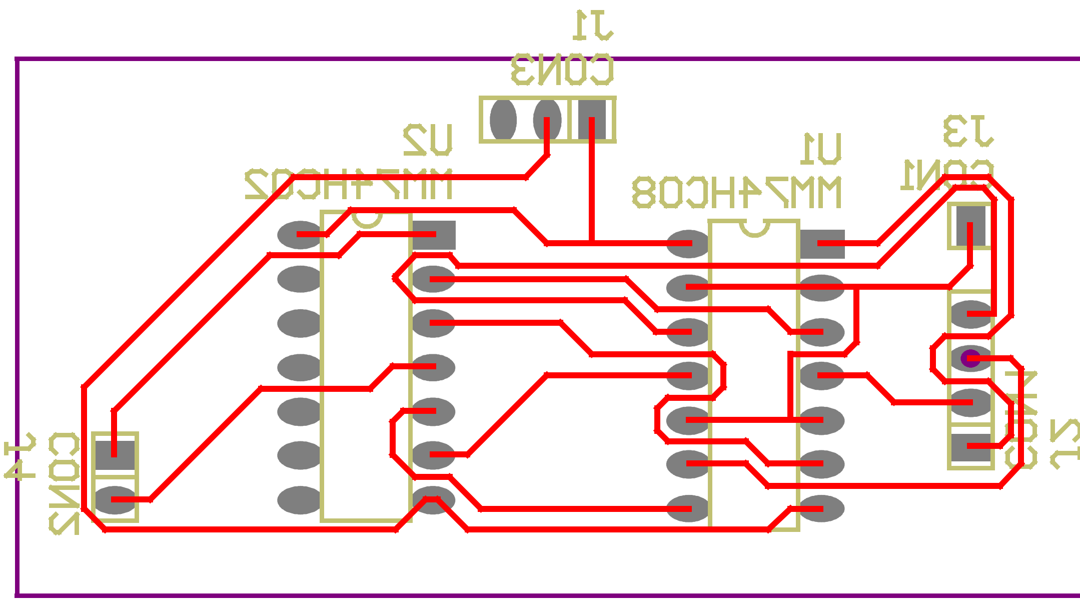
- We can first layout the components. Clicking a footprint and dragging the mouse will allow you to move the component. Pressing the Space Bar while doing this causes it to rotate. You should try to place the components reasonably close together (save surface area), and in a manner that ensures that the connections cross each other as little as possible. This will make it easier for Protel to route the design. For this design, you can use the figure on the next page as a reference.

- We now need to define the dimensions of the circuit board. This can be done by drawing a box around the components in the Keepout Layer. In the lower left of the screen there is a pull-down box that selects the currently layer you are working in. Ensure this is set to *KeepoutLayer*. Then use **Place/Line** to draw the box.

- We now need to tell Protel's AutoRouter to work only in the Top Layer. This can be done through **Design/Rules**. Select **Routing/Routing Layers**, click **Properties**, and set Bottom Layer to *Not Used*.

- We also should select **Routing/Width Constraint**, and change the parameters from *10mils* to *15mils*. This will cause the Router to create electrical traces .015” wide. This should be a minimum. Traces always benefit from being wider, especially Power Signals. If a high amount of current goes through a thin trace, it will blow. Therefore, If it looks like your board can fit larger traces, increase the width constraint.

- Now select **AutoRoute/All**, and hit **Route All**. The Board should Route, with the result being similar to the figure below.



- Now we can run a Design Rule Check (DRC), which will ensure that we have no short circuits, that everything is routed, etc. It can be accessed from **Tools/Design Rule Check**. Leave the Defaults set, and click **Run DRC**. A text file should appear and say that no violations appeared. If there are issues, fix them, use **Tools/Reset Error Markers**, and then rerun the DRC.

The PCB design is now complete. All that remains is to transfer the PCB out of Protel in a format that can be understood by either the PCB manufacture or the T-Tech software.

- Your board design should now be ready to be manufactured. This is accomplished by exporting the design to a series of files know as Gerber Files. These are an industry standard format that all the PCB manufactures use for layout information.

- Run **File/CAM Manager**. Select *Gerber* for the Output. Leave the name as it is, and ignore the notice about Embedded Apertures. Leave the Defaults for the Next Window (Units and Format). For the Gerber Layers to Plot, Select only the Top Layer and the Keepout Layer, and make sure neither is set to be mirrored. Leave the remaining settings as default.

- Protel Now displays a new window whose Tab is Titled Cam Outputs. Run **Edit/Insert NC Drill**. Use the Defaults and click **OK**. The main window should now say both *Gerber Output 1* and *NC Drill Output 1*. Run **Tools/Generate CAM Files**.

- Return to the **Documents Folder**, and then open the **CAM Folder**. You should see several files. We are interested in the *.GTL (Gerber Top Layer), *.GKO (Gerber Keepout Layer) and *.TXT (NC Drill) Files. Select these files and run **File/Export**. There should be a shortcut on the Desktop to *T-Tech Files.* If you cannot find it, ask the IMDL TA. Create a Folder with your name if it does not already exist, and then create a folder with the name of this design, and save the three files there.

- One more piece of information is required: return to your PCB Design. Use **Report/Measure Distance** to measure the dimensions of your Circuit Board. Save this information in a text file, and include it in the Design Folder in *T-Tech Files*. Then inform the IMDL TA that you are ready to have your PCB milled out.