

ELE1EDP Laboratory 2007: ALTIUM DESIGNER 6.0 Schematic Editor

1 Preparing the Folders

After you login, use Windows Explorer to enter **H:** drive (which is your own account's folder), then select **File -> New -> Folder**. Name this new folder **ELE1EDP**, enter it, then create a second new folder inside that one, named **Counter**.

2 Starting Altium Designer and Creating a New Design Database

On the Windows desktop, double-left-click on the **ALTIUM DESIGNER 6.0** icon. This should start **Design Explorer**. (Click **NO** to the UPDATES question box.)

Select **File -> New Project -> PCB PROJECT**.

Now there should be a project titled **PCB_Project1.PrjPCB** in the left hand side panel.

Select **FILE -> Save Project As**.

Save your new design database as **Counter.PrjPCB**.

Every five (5) minutes, keep saving your work!

3 Schematics

Right click on "**Counter.PrjPCB**" on the left hand side Project panel.

Select **Project -> Add New to Project -> Schematic**.

Choose **FILE -> Save All**.

When prompted for the Schematic File Name, type in "**Counter.SchDOC**".

Select **Design -> Document Options...**. Set **Standard Style** to **A4** to specify the correct paper size. Then left click **OK**.

The lower right corner of the schematic sheet contains a title block. Compare it with the example schematic. Select **Place -> Text String** (or type **PT**) to fill in the title block with the following information:

Title	COUNTER	
Number	EDP-1	
Revision	0	
Sheet	1	of 1
Drawn by:	<i>Your Name</i>	

4 Schematic Component Libraries

Most of the components available in **Altium 6.0** are stored in databases containing libraries arranged by manufacturer and category of product. Libraries have to be loaded before their components can be accessed.

The devices required for the Mini-Project include an LM555 timer, a 4040 counter, resistors, capacitors, light emitting diodes (LEDs), and a switch. These are found in three libraries:

Miscellaneous Devices.lib

Texas Instruments Analog Timer Circuit.INTLIB

Motorola Logic Family

In tables we'll abbreviate the names of these libraries as **Misc**, **Timer** and **Logic**.

Select **Design -> Add/Remove Library**, click **Install**, and select the following databases:

TEXAS INSTRUMENTS -> TI Analog Timer Circuit

Motorola -> Motorola Logic Gate

Motorola -> Motorola Logic Counter

Add them. This loads all the libraries contained in that database. Any libraries other than the ones described above, which you need, you may **Delete**.

5 Finding the Parts for the Schematic

In schematics the components are described by **Designator** and **Part Type**. The **Designator** is the unique name of the particular component. For example, two 10k resistors must have distinct designators (eg, R1 and R2) in order to distinguish them. **Part Type** means either the value or the operating function of the component, sufficiently specified to calculate its exact effect in the circuit.

The following table lists all the types of components in the Counter circuit, except that you must search the Altium libraries for a 4040 counter. To find a component, look for its **Library Reference** name. Resistors and capacitors have values; place the component's value in the Comment field.

Description	Designator	Library	Lib Ref	Comment	Footprint
Timer	TIMER	Timer	NE555P		P008
Counter	COUNTER	Logic	MC54HC4040J		620-09
Resistors	R1 etc	Misc	RES2	8k2 etc	AXIAL-0.4
Electrolytic Capacitor	C1	Misc	CAP POL1	10uF	RB7.6-15
Nonpolar Capacitor	C2	Misc	CAP	10nF	RAD-0.3
Light Emitting Diodes	H1 to H4	Misc	LED1		LED-1
Hex O/C Inverter	D1 to D4	Logic	MC74HC04N		646-06
Counter Reset switch	RESET	Misc	SW-SPST		SPST-2

6 Guidelines for Drawing your Schematic

VERY IMPORTANT:

- After every **five (5) minutes** of work, use **File -> Save All** to save your work in progress.
- Don't draw yet! **Wait** until you reach the section on **Placing the Parts**.

When placing components on the work sheet, conscientiously follow these guidelines:

- **Inputs** are on the **left**
- **Outputs** are on the **right**.
- Components that are directly **connected** are reasonably **close** to each other.
- The supply voltage must always be named **VCC**.
- Ground must always be named **GND**.

Useful commands:

- **One right-click** will place (or draw) the current object, but you will still be in the **drawing mode** for that type of object, so you can continue drawing more objects of that kind.
- **Two right-clicks** will **exit the current drawing mode**, to let you draw different kinds of objects.
- To **select** an object, **left-click** it.
- If two or more objects are selected when you tried to click on one, then a box will pop-up listing all the selected objects, one per text line. Proceed by clicking on the line describing the object you want.
- A more powerful means to select objects, is to use **Edit -> Select**. This provides several ways to select a **set** of objects.

OVER/

- To **move** an object, **left-click** on it, then **drag** it with the mouse.
- Another means to move objects uses **Edit -> Move**. This works with objects selected by **Edit -> Select**.
- If you need to **rotate** an object, first **select** the object, then press the **space bar** to perform the Rotation.
- To **deselect**, you can use **Edit -> Deselect -> All**.
- To **edit** an object, **double-left-click** on it. This should display a table listing its **designator**, **part type**, **library reference**, and **footprint**. You can ignore the other parameters.
- To make **copies** of an object, use **Edit -> Select -> Inside Area**, followed by **Edit -> Copy**, then use **Edit -> Paste** as many times as necessary. You will probably then need to change the **designator** or **part type** of each copy.
- To **delete** an object, select it by left-clicking on it once, then press the **Delete** key.
- In case of a mistaken deletion, select **Edit -> Undo**.

7 Placing the Parts

7.1 555 Timer

Select **Design -> Browse Library** and look for the component named **NE555P**. Press **OK**. Place the 555 timer toward the top left of the schematic sheet, as shown in the example.

Notice that you could immediately place additional 555 timers, if you wished. But we only need one, so **right-click** once or twice to escape that mode. This quick multiple copy capability will however be especially useful when placing resistors and LEDs.

Now **double-left-click** on the 555 block and edit its Designator so it becomes **Timer**, as instructed in the parts table.

7.2 4040 Counter

Place the **4040** counter (the heart of our circuit) in a similar fashion.

7.3 Resistors and Capacitors

Capacitors and resistors are available in the **Miscellaneous Devices** library. You must ensure that the **part type** has the correct value (**10k**, **0.01 uF**, etc, as appropriate). The **designators** must agree with the corresponding names (**C1**, **R5**, etc) in the example schematic.

Remember to ensure that the **Designator**, **Part Type**, and **Footprint** of every Capacitor and Resistor is in accord with the parts table. If you **edit** the **first** resistor **before** placing any others, then the **subsequent** resistors will **inherit** the correct values.

7.4 LED circuits

As the parts table indicates, light emitting diodes can be found in the Miscellaneous library under the name LED. Look at the example schematic, and note that designators for LEDs commence with the letter **H**. This is in accord with an international standard which will be described in later lectures.

If you place the LED **H1** first, and edit it according to the parts table, so that its Designator reads **H1**, its Part Type **LED**, and its Footprint **DIODE0.4**, then place **H2**, **H3** and **H4** in that order, then their Designator numbers will increment automatically, and their Part Types and Footprints will all be correct.

The LEDs need drivers and current-limiters. Place a (different) 330 ohm resistor in series with each LED to limit the current.

For drivers choose the 74LS05 open-collector inverter integrated circuit: it contains six (6) inverter gates. Use four (4) of these gates, each to drive one LED.

7.5 Power Ports

For VCC and GND, use **Place -> Power Port**. Select the **VCC power port** and the **GND power port**.

Alternatively, VCC and GND can be accessed from the **Quick Links** Tool Bar.

7.6 Wires

Select **Place -> Wire**. Rest the mouse pointer on the first wiring symbol.

Wires can be connected when a red cross appears over the mouse cross-hair.

Notice that you can continue a wire at a different angle by pressing **left-click**, then continuing to draw. Please draw **vertical** and **horizontal** wire segments.

Clicking on a wire can allow you to adjust its corners and endpoints.

7.7 Junctions

Where wires join, Altium Designer automatically places **junctions**. Usually it does so correctly, but you can delete junctions like any other object, and place them from the appropriate toolbar (an exercise for the reader!), or by typing **PJ**.

8 Electrical Rule Check

When you consider that your circuit is complete, select **Projects -> Compile Document -> Counter.SchDOC** to confirm that the circuit schematic is electrically valid.

In case of error, *especially* look for **wires hidden** under other wires, for **wires that extend too far**, or **not far enough** to make contact with pins, wires that pass by the **ends of pins**, and any *suspicious* or *unexpected* **junction**.

9 Net List

A **net list** is a text description of the schematic, listing the components and their connecting nodes. Each **component** is described by several items on separate lines between square brackets, and every **node** is described by a list of component pin numbers between round brackets.

Select **Design -> Netlist for Project -> Protel**.

View the Netlist file by selecting **Generated -> Netlist Files -> Counter.NET**.

What are the meanings of the three (3) items listed for each component? In what order are they listed?

Compare the **net list** with the circuit components and connections in the **original** example schematic to **check that you have not made any mistakes or omissions** in your own schematic.

If there are any discrepancies, then edit your schematic until it is fixed.

10 Shut Down

Before even thinking of shutting down the computer, you should do all of the following.

1. Repaired all problems so that your project has no errors when compiled.
2. Obtain a Net List that a demonstrator confirms is correct.
3. Save all your work (**File -> Save All**).
4. Obtained all 100 marks from the demonstrators.

Only then may you proceed to shut down, as follows.

1. Exit the Altium Designer by selecting **File -> Exit**.
2. Click on the MS-Windows **Start** button.
3. Select **Shut Down -> Shut down the computer**.
4. After a minute (**be patient!**), you will be told that you can switch off the computer.
5. Do so.

Geoffrey Tobin and Adam Console

Version 14

Sunday 29 July 2007

OVER/

