

ELE1EDP: Electronic Design Project

2007 Laboratory:

PSpice: Introduction

1 What is PSpice?

Spice is a circuit simulation tool, widely used around the world for decades. PSpice is the PC version of Spice. You will be using a free evaluation copy of a Microsoft Windows version of PSpice.

Circuits to be simulated are entered into a schematic editor (a little reminiscent of the one in Protel).

2 Starting

- **Login** to a computer.
- Go to **H:** drive.
- Enter the **ELE1EDP** folder.
- Create a **subfolder** and name it **PSpice**.
- Start the PSpice **Schematics** program.

3 Shortcuts

- **Ctrl + G** = Draw -> Get New Part.
- **Ctrl + M** = Marker -> Mark Voltage/Level.
- **Ctrl + P** = Place Part.
- **Ctrl + R** = Rotate.
- **Ctrl + W** = Draw -> Wire.

4 Battery and Resistor Simulation

Open a new schematic: **File -> New**.

Save the circuit file as follows.

File->Save As.

Navigate your way to the **H:\ELE1EDP\PSpice** directory.

Name your circuit file: **RES**

Click on **Save**.

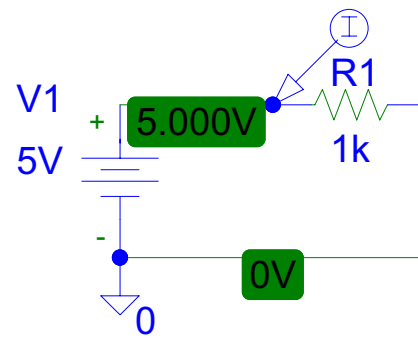
The file created should be named **RES.sch**.

If you haven't saved the circuit, an asterisk * will be present to the left of the file name in the file's title.

Please be aware that PSpice schematics are **not** compatible with Protel schematics. The common suffix "**sch**" therefore requires that you separate PSpice circuits from Protel schematics by placing them in distinct directories, as instructed.

Now enter the battery and resistor circuit given in the first diagram, as follows.

B



Select the **ANALOG** library.

Ctrl + G, select the part by typing its name, Place, LeftClick.

Here is the parts list for this circuit.

Part	Quantity
R	1
VDC	1
GND_ANALOG	1

Save this schematic. (Regularly!)

4.1 Hints

- Left Click on a part to highlight it.
- If you wish to delete a part: highlight it, then press the Delete key.
- In case of error, examine the PSpice output text file by: **Analysis -> Examine Output**.

4.2 DC Sweep

Use a current marker to measure the current flowing into the resistor R1:

Markers -> Mark Current into Pin.

Place the marker at the junction between the battery and the resistor.

To simulate the behavior of the 1K resistor as the battery voltage increases from 0V to 5V in steps of 1V (this is called a **DC sweep**), do this:

1. **Analysis -> Setup**
2. Enable DC Sweep by clicking on the check box (a tick should appear).
3. Press the **DC Sweep** button.
4. Enter these four parameter values:

Parameter	Value
Name	V1
Start Value	0
End Value	5
Increment	1

Note: You can **Tab** to the next text box, and **Shift+Tab** to the previous one.

Press **OK**, then press **Close**.

Press the **F11** key to start the simulation. (Equivalently, select **Analysis -> Simulate**.)

MicroSim Probe should soon display a graph of the current **I(R1)** versus the voltage of the battery **V_V1**.

Note: If the current is negative, move the battery to the left (by highlighting it, then dragging the mouse). This will create a wire. Place the current marker on the wire. Alternatively, move the current marker to one pin of the resistor.

Have a demonstrator check your result for the DC sweep.

File -> Save (or **Ctrl+S**).

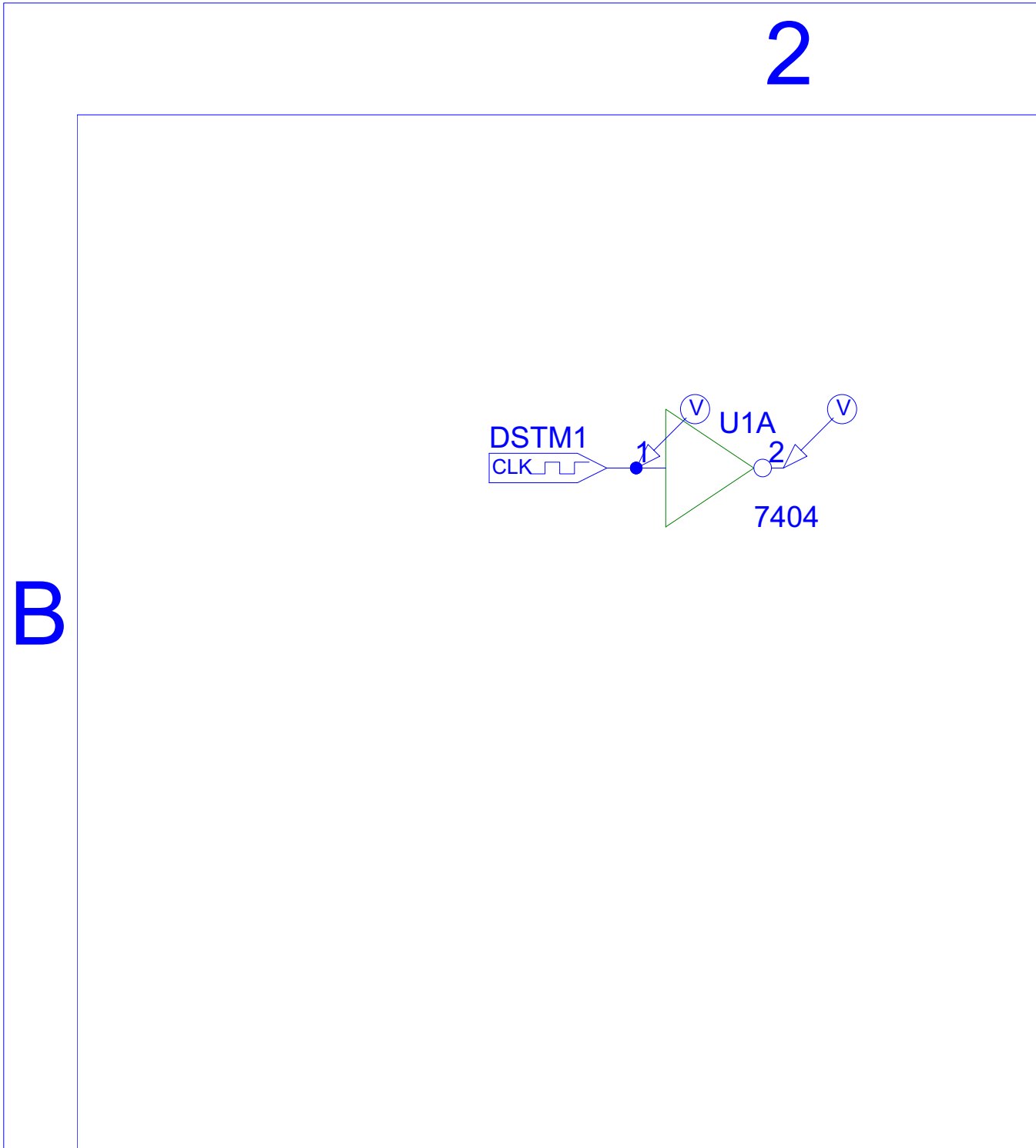
File -> Close.

5 NOT Gate

PSpice also simulates digital circuits.

Open a **New Schematic**, save it under the name **NOT**.

Draw the schematic given in the second diagram.



Part	Quantity
DigClock	1
7404	1

Double click on the Digital Clock, DSTM1, and enter the following parameters. After entering each value in the text box, click **Save Attr**.

Parameter	Value
DELAY	0
ONTIME	0.5s
OFFTIME	0.5s

OK.

Place a voltage marker (**Ctrl+M** or **Markers -> Mark Voltage/Level**) on the input, and another on the output, of the NOT gate, **U1A**. (The "A" means the first element of the IC package U1.)

Simulate the effect of the square wave and the response of the gate at the two voltage markers, by using **Transient Analysis**.

1. **Analysis -> Setup**
2. *Disable* **Bias Point Detail**
3. Enable **Transient**
4. Click on **Transient**
5. Enter these essential parameters:

Parameter	Value
Print Step	0.5s
Final Time	5s

Ignore the other parameters.

OK.

Close.

F11.

MicroSim Probe should produce a plot of **DSTIM1:1** and **U1A:A** versus **time**.

This result must also be checked by a demonstrator.

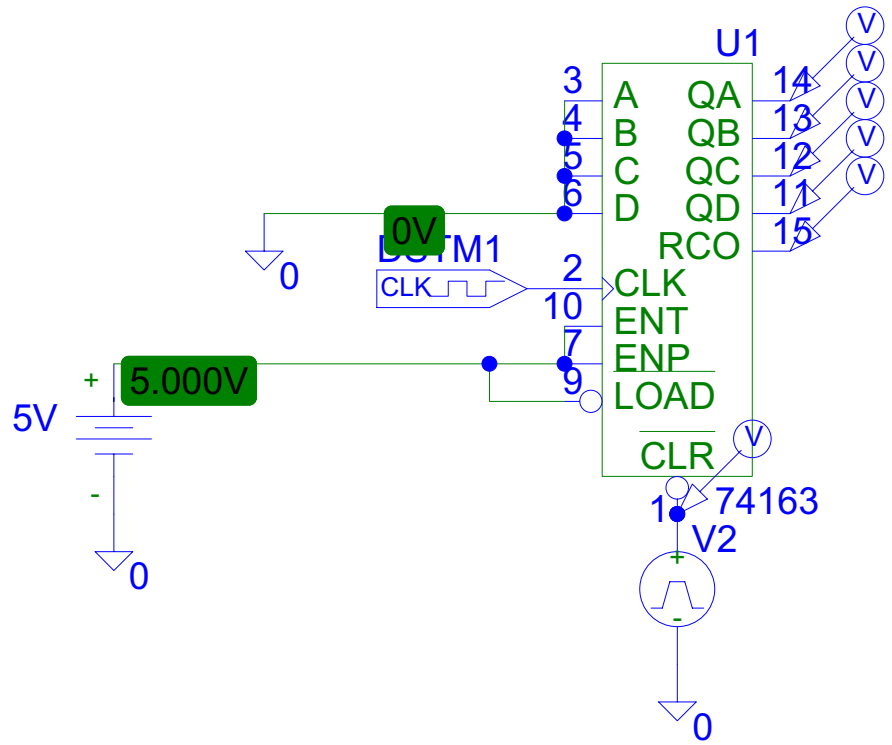
6 Counter

Mixed analog and digital circuits can be simulated.

Open a New schematic, called **COUNTER**.

Draw the schematic of the third diagram.

B



The parts are:

Part	Quantity
74163	1
DigClock	1
VDC	1
VPULSE	1
GND_ANALOG	3

The Digital Clock has these parameters:

Parameter	Value
DELAY	0
ONTIME	0.5s
OFFTIME	0.5s

For the Voltage Pulse:

Parameter	Value	Comment
DC	0	DC Offset
AC	0	Superposed Sinusoid
V1	5V	Resting Voltage
V2	0	Pulse Voltage
TD	0	Delay Time
TR	1ms	Rise Time
TF	1ms	Fall Time
PW	1s	Pulse Width
PER	100s	Pulse Repetition Period

The Battery must have DC = 5V.

The Analog Grounds must be 0 V.

Set Voltage Markers (**Ctrl + M**) at each of these inputs: CLK and CLR; and each of these outputs: QA, QB, QC, QD, and RCO.

Note: The 74163 is a fully synchronous 4-bit counter. RCO is its Carry Out signal, which goes high when all of QA to QD are high, in preparation for the next clock pulse which will “increment” the counter to zero.

Select **Analysis -> Setup -> Transient** with these parameters:

Parameter	Value
Print Step	0.5s
Final Time	20s

Press **F11**.

The Probe graph should include waveforms from all seven voltage markers.

Ensure that a demonstrator sees the output and marks your work.

Geoffrey Tobin
Version 5
Monday 20 August 2007