LAB 1 TUTORIAL

TIMER-COUNTER-7 SEG DISPLAY PROJECT

PREREQUISITES

Please complete the following steps before starting the lab:

1. Install and activate NI Multisim 14.2 by following the instructions in *Multisim-DesktopWindowsInstallGuide.pdf* on D2L.
2. Familiarize yourself with NI Multisim 14.2 by completing the tutorial (pages 1 to 17 only) in *Multisim-Tutorial.pdf* on D2L.
3. Read the Lab 1 Manual.

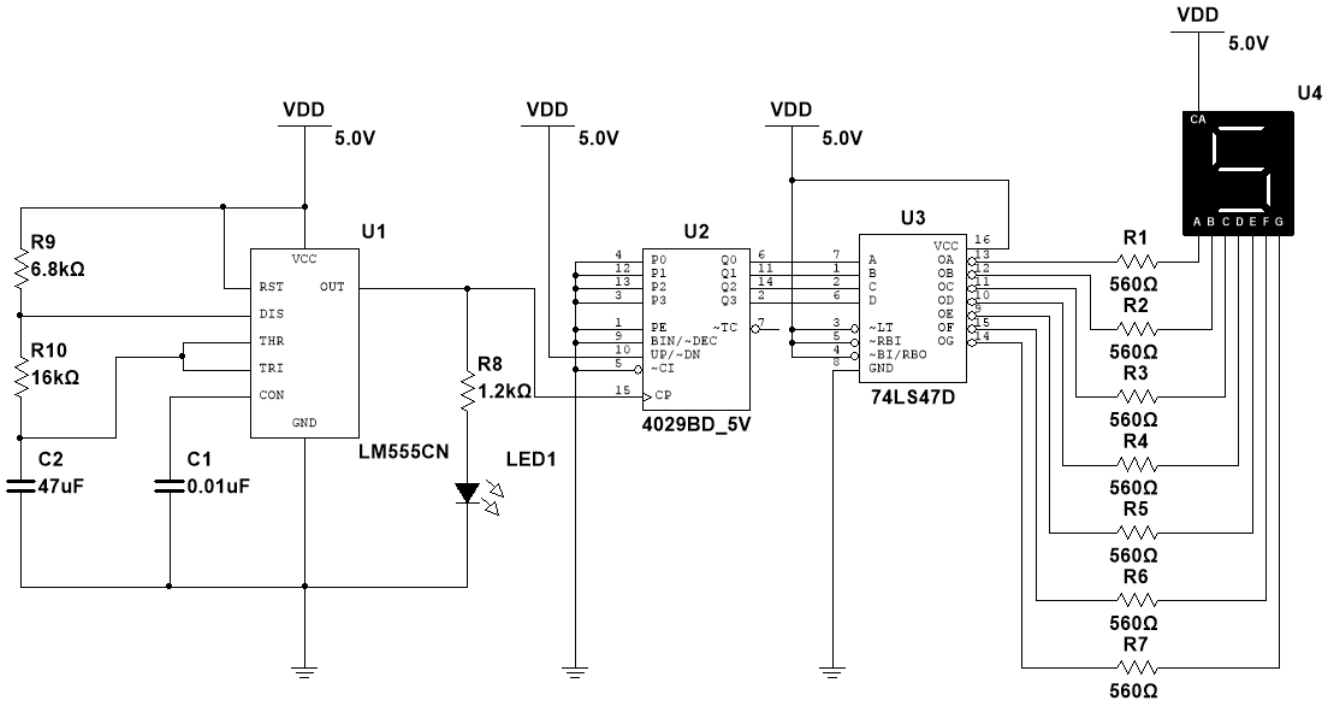

Figure 1

OVERVIEW

In this tutorial we will use NI Multisim 14.2 to:

1. Construct the circuit in the schematic diagram shown in Figure 1.
2. Measure the output frequency of the 555 timer.
3. Measure the outputs Q0, Q1, Q2, and Q3 of the 4029 Counter.
4. Find the inputs of A, B, C, D, E, F, and G corresponding to display values of 0 to 9.
5. Change the value of RA and measure the frequency.
6. Change the counter connection to countdown.

PROCEDURE

1. Schematic Design

Please follow the following instructions to construct the circuit shown in Figure 1:

1. Create a new folder “BME328” in your working directory (e.g. Desktop).
2. Create a new folder “Lab1” in your “BME328” folder.
3. Open NI Multisim 14.2.

To add a component right click on the white canvas and select “place component.” A window titled “Place a Component” will pop-up as shown in Figure 2.

Search for the following components in the “Place a Component” window and click the “OK” button to add them to your canvas.

1. RESISTOR_RATED (you need 9 of these).
2. CAPACITOR_RATED (you need 2 of these).
3. LM555CN.
4. LED_red.
5. 4029BD_5V.
6. 74LS474D.
7. SEVEN_SEG_COM_A.
8. VDD.
9. GROUND.

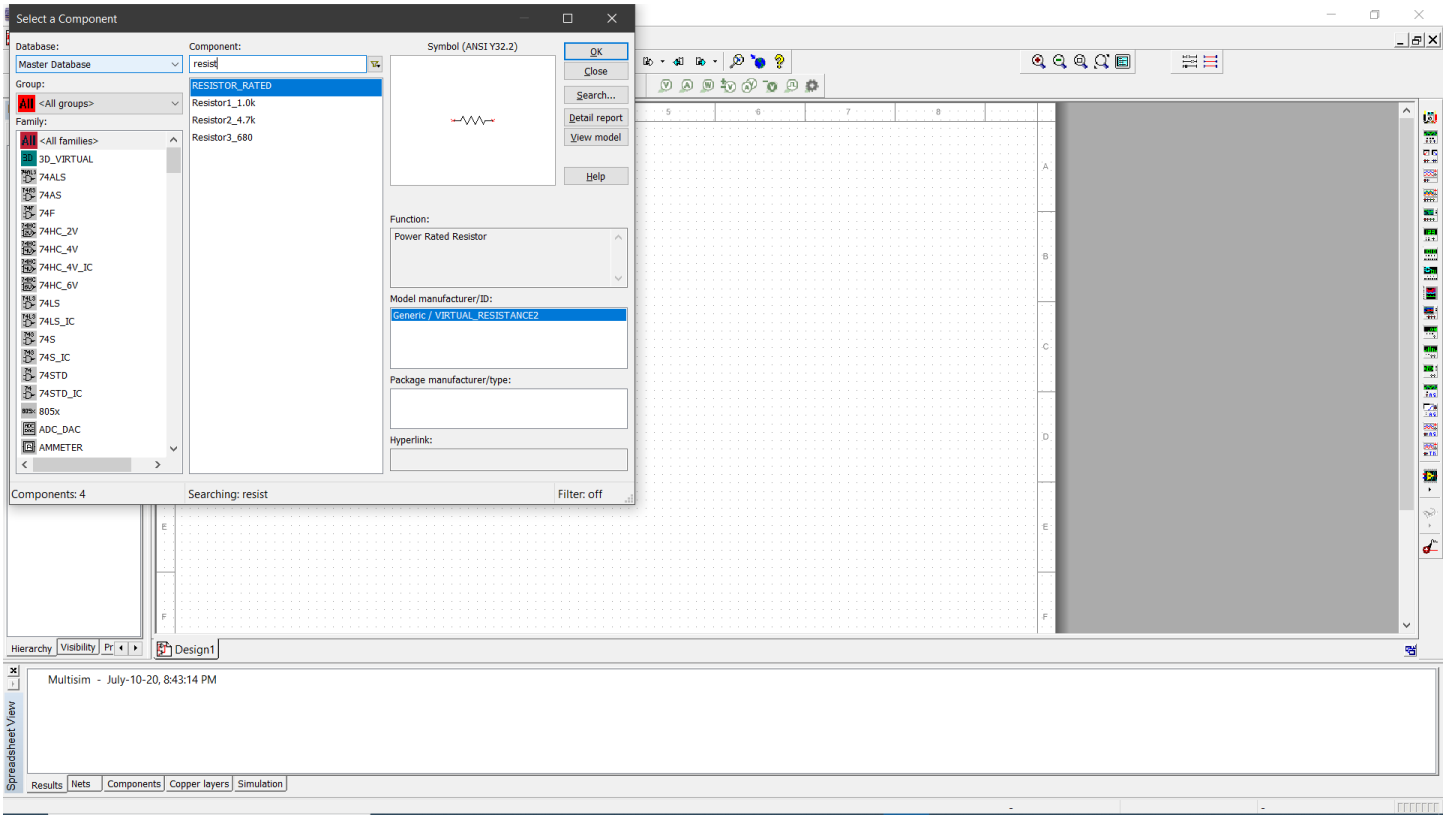
Select a Component
Database:
Master Database
Component:
resist
Symbol (ANSI Y32.2)
OK
Close
Search...
Detail report
View model
Help
Group:
<All groups>
Family:
<All families>
3D_VIRTUAL
74ALS
74AS
74F
74HC_2V
74HC_4V
74HC_4V_IC
74HC_6V
74LS
74LS_IC
74S
74S_IC
74STD
74STD_IC
805x
ADC_DAC
AMMETER
RESISTOR_RATED
Resistor1_1.0k
Resistor2_4.7k
Resistor3_680
Function:
Power Rated Resistor
Model manufacturer/ID:
Generic / VIRTUAL_RESISTANCE2
Package manufacturer/type:
Hyperlink:
Components: 4
Searching: resist
Filter: off
Hierarchy
Visibility
Design1
Multisim - July-10-20, 8:43:14 PM
Spreadsheet View
Results
Nets
Components
Copper layers
Simulation

Figure 2

Arrange the components as a shown in Figure 3.

Tips:

1. To flip or rotate a component, simply right click on the component and choose Flip or Rotate.
2. To change the name of a component:
   - Right click on the component, choose “Properties.”
   - Choose the “Label” tab at the top of the window that pops up.
   - Change the text field below the “RefDes” label to the new component name (e.g. R1).
3. To change the value of a component (resistance, capacitance, etc...)
   - Right click on the component, choose “Properties.”
   - Choose the “Value” tab at the top of the window the pops up.
   - Change the text field beside the “Resistance” or “Capacitance” labels to the new value (e.g. 6.8k,16k, 47u, 0.01u).
4. To draw a wire:
   - Move the mouse cursor to the edge of one of the pins that you want to connect.
   - When the cursor changes to a bubble form, click on the edge of the pin you want to connect then click on the edge of the other pin that you want to connect.

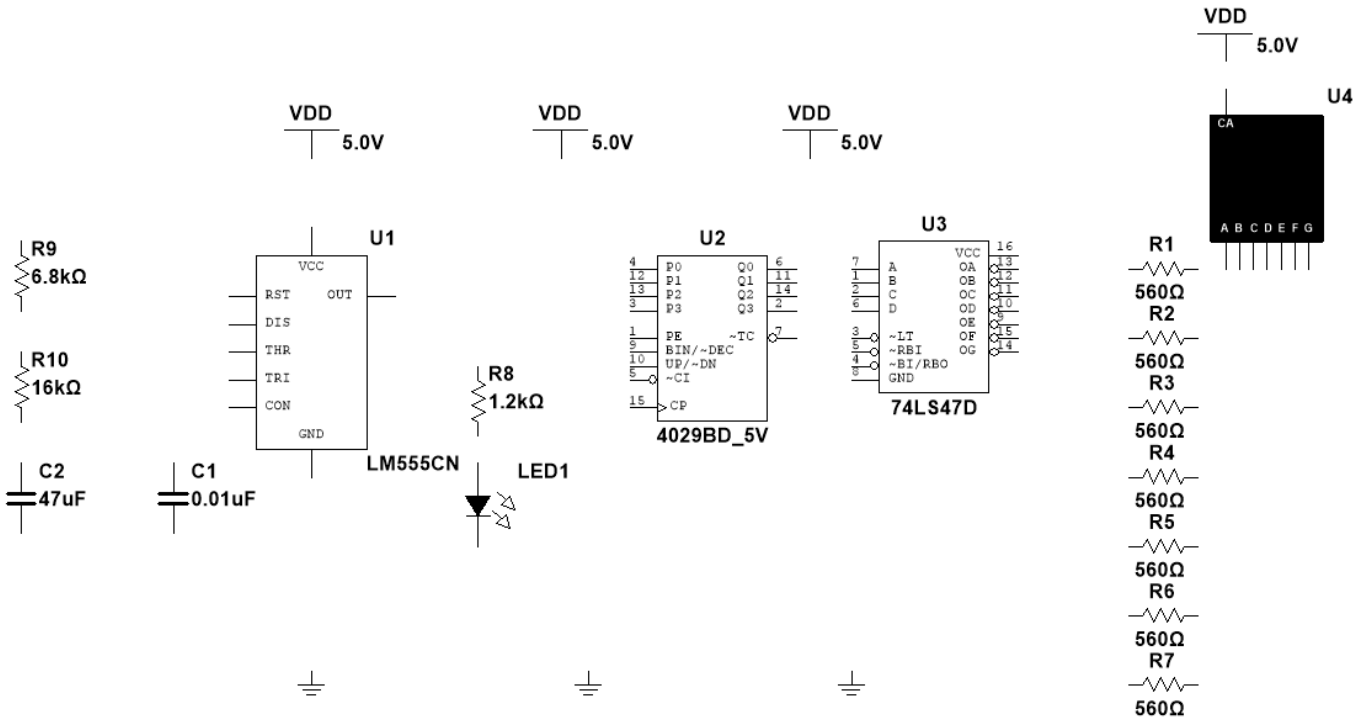
VDD
5.0V
VDD
5.0V
VDD
5.0V
VDD
5.0V
U4
CA
A B C D E F G
R1
560Ω
R2
560Ω
R3
560Ω
R4
560Ω
R5
560Ω
R6
560Ω
R7
560Ω
R9
6.8kΩ
R10
16kΩ
C2
47uF
U1
VCC
RST
OUT
DIS
THR
TRI
CON
GND
LM555CN
C1
0.01uF
R8
1.2kΩ
LED1
U2
P0
P1
P2
P3
Q0
Q1
Q2
Q3
PE
~TC
BIN/~DEC
UP/~DN
~CI
CP
4029BD_5V
U3
VCC
A
B
C
D
OA
OB
OC
OD
OE
~LT
~RBI
~BI/RBO
GND
OF
OG
74LS47D

Figure 3

Connect the components as a shown in Figure 4.

Tip:

- To draw a wire:
  - Move the mouse cursor to the edge of one of the pins that you want to connect.
  - When the cursor changes to a bubble form, click on the edge of the pin you want to connect then click on the edge of the other pin that you want to connect.

Save your design as “lab1.ms14” in your “Lab1” folder.

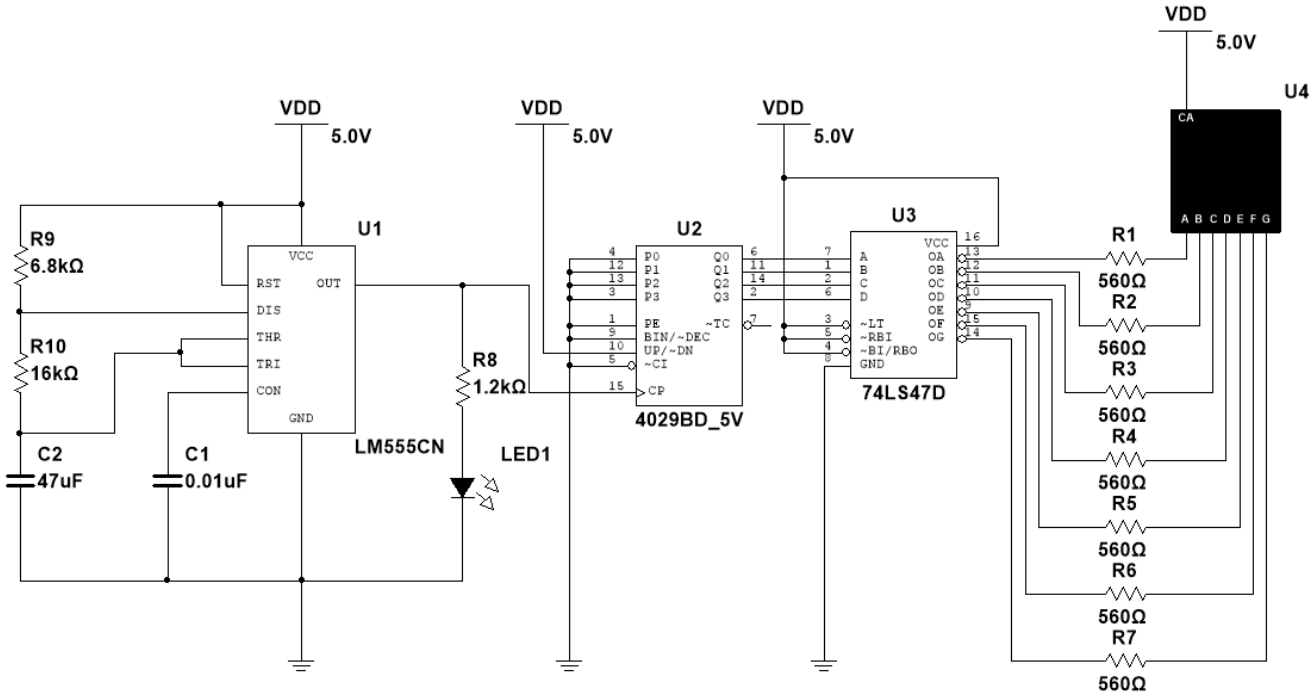

Figure 4

2. Simulations

Before we simulate, we need to label the nets that we wish to simulate. To label a net:

1. Double click on each of the wires highlighted in Figure 5.
2. Then change the “Preferred net name” field as illustrated in Figure 4 (i.e Vout555, etc…)

Save your design as “lab1” in your “Lab1” folder.

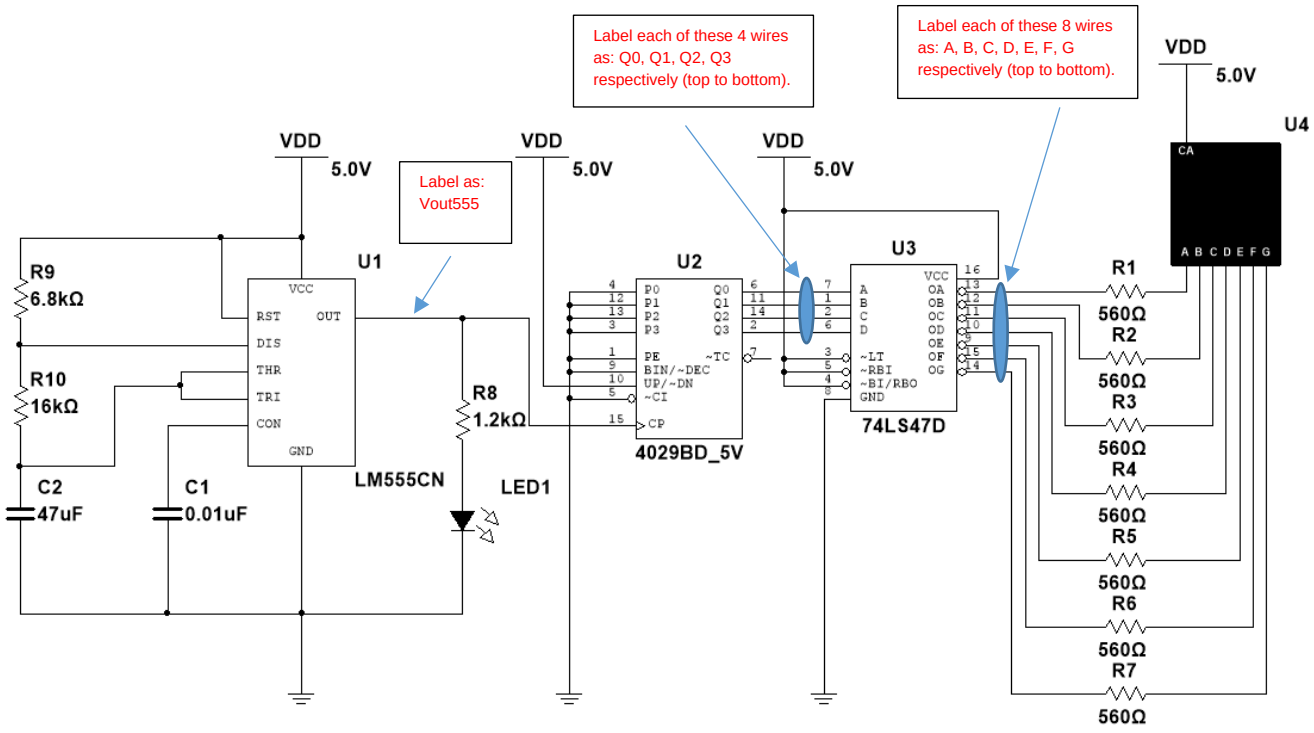

Figure 5

To simulate the design:

1. Click on “Simulate” located at the top left of your Multisim window.
2. Then click on “Analyses and Simulation.”
3. Follow steps 1 to 7 outlined in Figure 6 and Figure 7.
4. Compare your results to Figure 8.

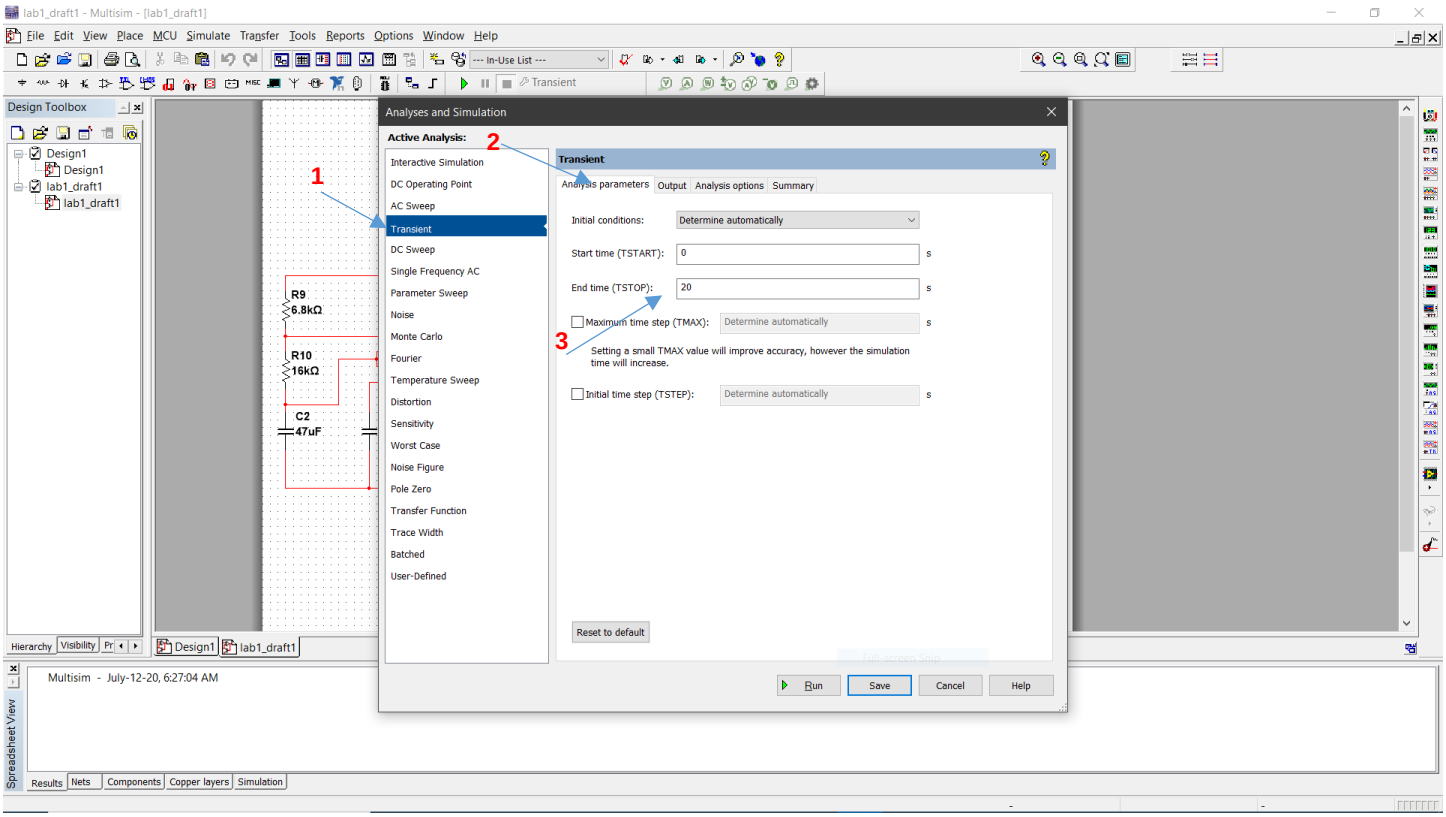

Figure 6

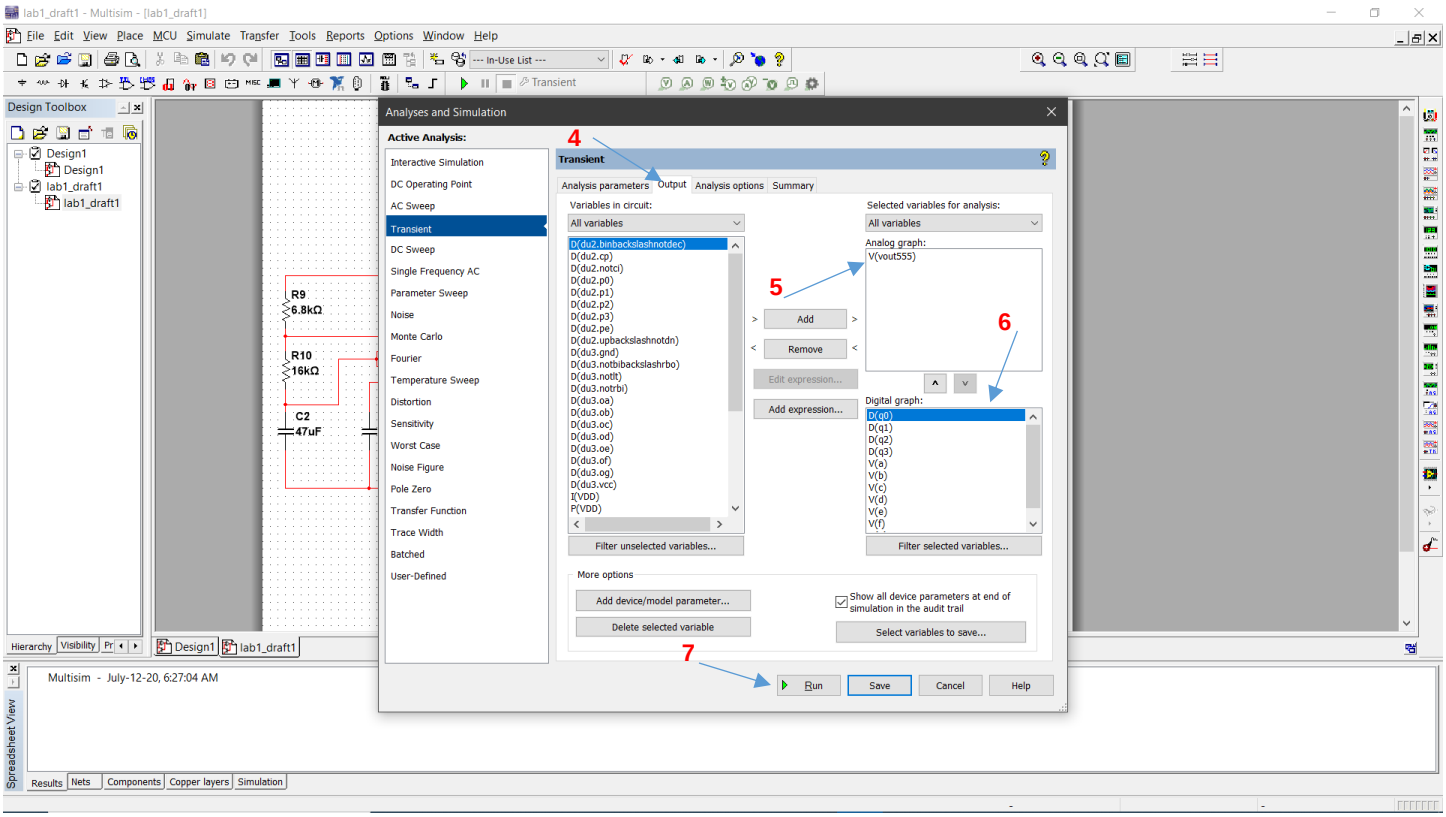
lab1_draft1 - Multisim - [lab1_draft1]
File Edit View Place MCU Simulate Transfer Tools Reports Options Window Help
Design Toolbox
Design1
lab1_draft1
R9
6.8kΩ
R10
16kΩ
C2
47uF
Analyses and Simulation
Active Analysis:
Interactive Simulation
DC Operating Point
AC Sweep
Transient
DC Sweep
Single Frequency AC
Parameter Sweep
Noise
Monte Carlo
Fourier
Temperature Sweep
Distortion
Sensitivity
Worst Case
Noise Figure
Pole Zero
Transfer Function
Trace Width
Batched
User-Defined
4
Transient
Analysis parameters
Output
Analysis options
Summary
Variables in circuit:
All variables
Selected variables for analysis:
Analog graph:
V(vout555)
5
Add
Remove
Edit expression...
Add expression...
6
Digital graph:
Filter unselected variables...
Filter selected variables...
More options
Add device/model parameter...
Delete selected variable
Show all device parameters at end of simulation in the audit trail
Select variables to save...
7
Run
Save
Cancel
Help
Multisim - July-12-20, 6:27:04 AM
Spreadsheet View
Results
Nets
Components
Copper layers
Simulation

Figure 7

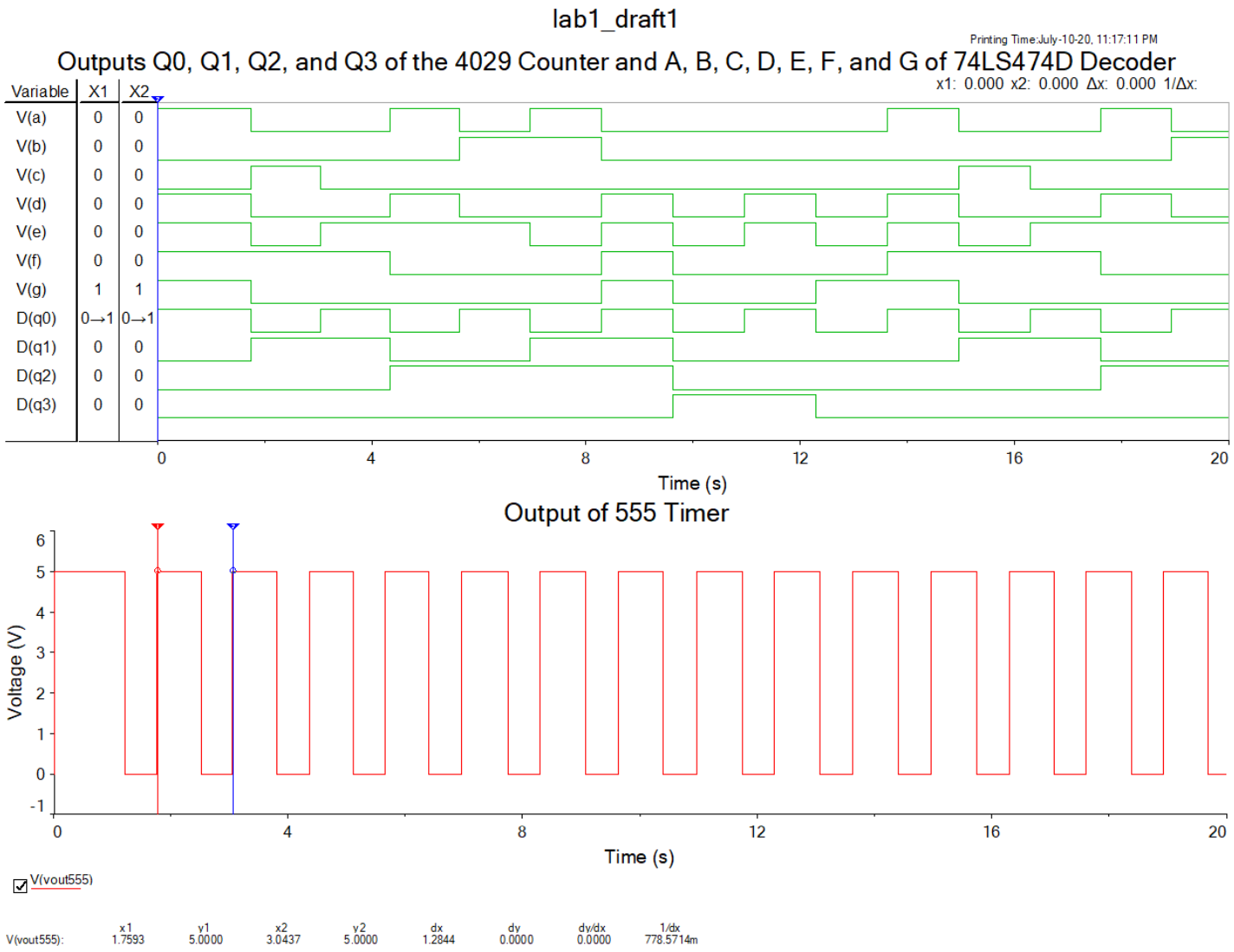

Figure 8

Frequency of the 555 Timer = 1/1.287516s = 0.777 Hz