ETEC644 Project

Using Node Listing for Electronic Circuits as a Teaching Tool

Nita Leighton

Contact information

designer@g-cube.com

Deadline May 30, 2014

Instructions for Node Layout and Node-List project for Beta Testers

By Nita Leighton

Welcome everyone who has volunteered to help on the Node Layout and Node-List project as Beta Testers. Thank you one and all for taking time out of your busy schedules to help me complete this graduate project this spring.

There are eight students, and a husband of a student, who was volunteered, to participate. Some of you have done this node layout/listing in previous classes with me, or another instructor. Others of you have not been exposed to this knowledge until now. This makes for a great mix of volunteers. Again, thanks.

In this packet are the following items: (except PPTs)

1 - Pre Quiz on Basic Electronic Symbols --**DO THIS FIRST**

2 - Materials list and instruction for node layout and node-listing (**Read** immediately)

****Two PowerPoint (PPT) presentations** (available on instructor's website www.classjump.com/Leighton) - Look for "grad projects" category

PPT 1 - Learning Basic ELEC Symbols (**Review** after taking pre quiz)

PPT 2 - How to do a node-layout (**Review** prior to doing node-layout / listing)

3 - Node-listing consolidation requirements (**Read** before starting node layout/listing)

4 - Node-list form (4 copies) (for node-listing)

5 - Drawing 1 -- Simple Starter Circuit (SSC) Drawing (for node-layout)

6 - Drawing 2 -- Comparator Schematic Drawing (for node-layout)

7 - Drawing 3 -- Transistor Intercom Schematic Drawing (for node-layout)

8 - Post Quiz on Basic Electronic Symbols (taken when all node layouts / listings done

9 - Evaluation (Exit survey)

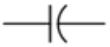
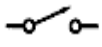
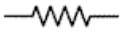
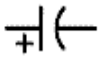
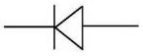
10 - If done completely on-line, please email completed materials to instructor by May 30, 2014

ADDITIONAL Copies of the drawings, node listing form, quiz and evaluation will also be on website

PRE / POST **Basic Electronic Symbol Quiz** **Name:_____**

First, write your name on the quiz. Second, circle whether this is a pre or post quiz.

The object of this quiz is to see if the learner can identify the common electronic symbols, used in drafting and printed circuit board design layouts. Match the component symbol with the component name/number, by filling-in-the-blank (right column) with the number found next to that name in the center of the page. Then, you are to fill-in-the-blank (left column), with the same number, for the appropriate common letter designation used in industry for that component. The components symbols on the quiz are the same type that will be found on the schematic drawings, which will be used in the node layout. The letter(s) designators are those that will be found on the schematic drawings and used on the node-listing form.

GND	____	1 - Battery (multi cell)	____	
J	____	2 - Capacitor (mica, stack, non-polarized)	____	
K	____	3 - Capacitor polarized	____	
Q	____	4 - Diode	____	
R	____	5 - Ground	____	
C	____	6 - Jack	____	
R	____	7 - Plug	____	
SW	____	8 - Microphone	____	
P	____	9 - Potentiometer	____	
R	____	10 - Push Switch	____	
B	____	11 - Resistor	____	
S	____	12 - Speaker	____	
D	____	13 - Switch (SPST)	____	
C	____	14 - Thermistor	____	
M	____	15 - Transistor (NPN)	____	

Materials for node-layout and node-listing project

If the node layout is done by hand, the following items are highly recommended:

1 - #2 pencil - sharpened

1 - good art eraser (art gum) or pink eraser, clean, if possible

1 - 6" or 12" ruler, transparent is better than solid, a triangle will work too

1 - Set of colored pencils with 10 colors if possible, 99cent store carries them

It is much easier and faster to do this work in regular pencil, and color pencil, than in ink or with markers. If there is a mistake, omission or change that needs to be done, then it is easier to correct in pencil.

Markers have a more limited color palette than the colored pencils. For the node listing place the color in the small rectangle provided on the node-listing form, and write in the color name to the right. When you write the node and the node number on the schematic, it really is better to use regular lead pencil. Some pale colors, like yellow are hard to read, if you want to underline the node number in that color that is fine to help identify it. The lines for the each node are to be done in one color. Each node has a unique color from its neighbor.

On the computer

The lines for the node layout can be done in several computer programs. It is time to experiment. It can be done in Word, PowerPoint, and Paint to name a few. In PowerPoint:

1 - Bring in the schematic drawing and insert it into a slide

2 - Under - **INSERT**

3 - Go to **SHAPES**- pull down and select the line tool (it will be in 1pt)

4 Position the line on one side of the wire of the circuit and stretch it to go as far as you need without crossing wires or other components, you will have to cross over the wire by the component, you will need to end the encircling back where you started. This makes a complete loop, so to speak.

5 - Make sure the lines are straight, parallel

6 - Go back **HOME** and the go to the right to DRAWING (drop down menu)

7 - Under **DRAWING** select line color, to change to the color you want (about 64 available)

8 - Under **DRAWING** select line style to change the thickness, to make line thicker (maybe use 2 or 3 point size)

9 - - May use copy and paste to repeat lines, saves time

Node-listing consolidation requirements:

A **constituent** is a uniquely identified component lead that is a member of a node group. It may be a wire on a part, an integrated chip (IC), ground, or whatever wire, that makes up an electronic circuit.

A node-list **MUST** be consolidated if there are redundancies within it. This reduction of redundancies will reduce the number of constituents in at least one node, or perhaps, may even reduce the number of total nodes. For example, ground is repeated in a circuit, but there is only one ground, which is attached to a number of different components, depending on the layout of the circuit.

Anywhere that the same node constituent appears in the same node or where the same constituent name appears in more than one node, then a consolidation **MUST** be performed.

Here are examples of where these redundancies may occur—where a power-supply point is shown in many places on the same schematic, such as identical ground symbols, +Vcc, +5V, -12V, +Vdd, -Vdd, or –Vee. Whenever these, or similar power supply points, are named in multiple places on a schematic, the node-listing rules will have made those points appear in multiple places. Originally, that would have been drawn that way to help to simplify the schematic and make it easier to draw and to read. But, remember that those identical names represent just ONE location on the power supply, so they are actually the same point. Any single node that calls out this point, more than once, must have the redundant, extra instances of that name, struck from that node, so that only one of those names is retained. Also, if a power supply point is found in multiple nodes, then those nodes must be consolidated into a single, larger node, because they are all hooked together at the power supply, even if they seem to be separate.

Node Location on Page--Legend -- Stay consistent on this -- USE Capitol LETTERS ONLY

Top =	T	Left =	L
Middle =	M	Center =	C
Bottom =	B	Right =	R

The example below shows how each landscape oriented schematic is broken up into zones of proximity. Some nodes may overlap the zone boundaries. If that happens, indicate the left-most or top-most zone only, for initial zone.

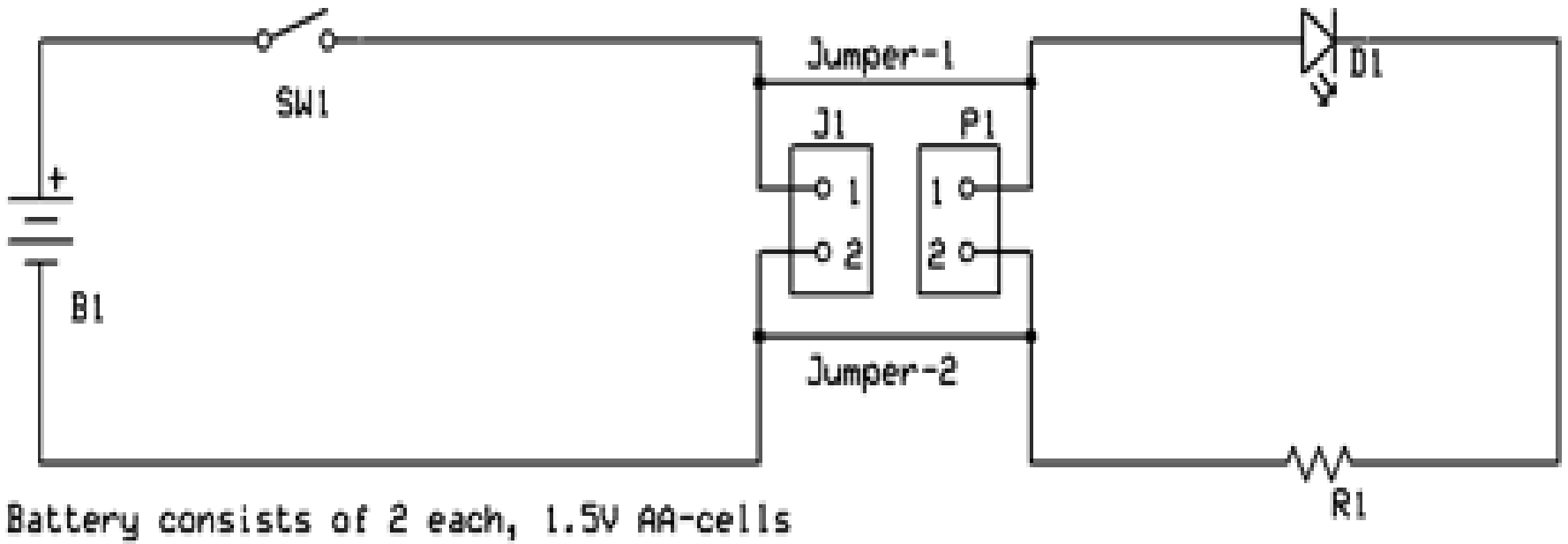
T L	T C	T R
M L	M C	M R
B L	B C	B R

The best approach for node-listing is to start in the top, left corner of the schematic drawing, and go left to right, and top to bottom, as you scan across it, encircling and documenting node-groups, using the same order in which you might read a written page. This is arbitrary, of course, because the order in which you document node-groups is somewhat irrelevant, except for conformity sake. As long as every node is documented, then the job is complete. As part of this node-group documentation, you will be encircling each node-group (aka: STRING of wires in a connection). You will plainly see which nodes are not documented when you look at your schematic and see any connected group that has not been encircled.

Remember to show the polarity of components, and the given-names of their wire leads, if they already exist, so the viewer can know which wire is being documented. Use the + or - signs, and the common letter/number designations associated with that particular component, using a one-up numbering sequence, as is usual on schematics. On the circuit, if only one polarity is shown, add the other to clarify it for the viewer. **If no markings exist on a multi-leaded component**, then add a unique letter of the alphabet to each of those wires on that component, until each has a letter on it. There will be a few components that have more than two wires. Stay consistent with all the lettering, all the way across the circuit (See example). You may repeat the letter of the alphabet on different components, so long as you include the component designator, along with the letter, or number, of each component pin. Remember, that there are Many, Many ways to do this--not just one. The only rule is that each and every wire lead, throughout the entire schematic, must have a uniquely identifiable designation on it, that cannot be confused with any other, anywhere in that schematic. You could even call every wire by a different person's name, if you had that many names to give. Whatever system you choose to use, just be consistent when you name or number them. And, if you have no strong preference as to what to call them, please use the suggestions that have been included, above, without getting fancy.

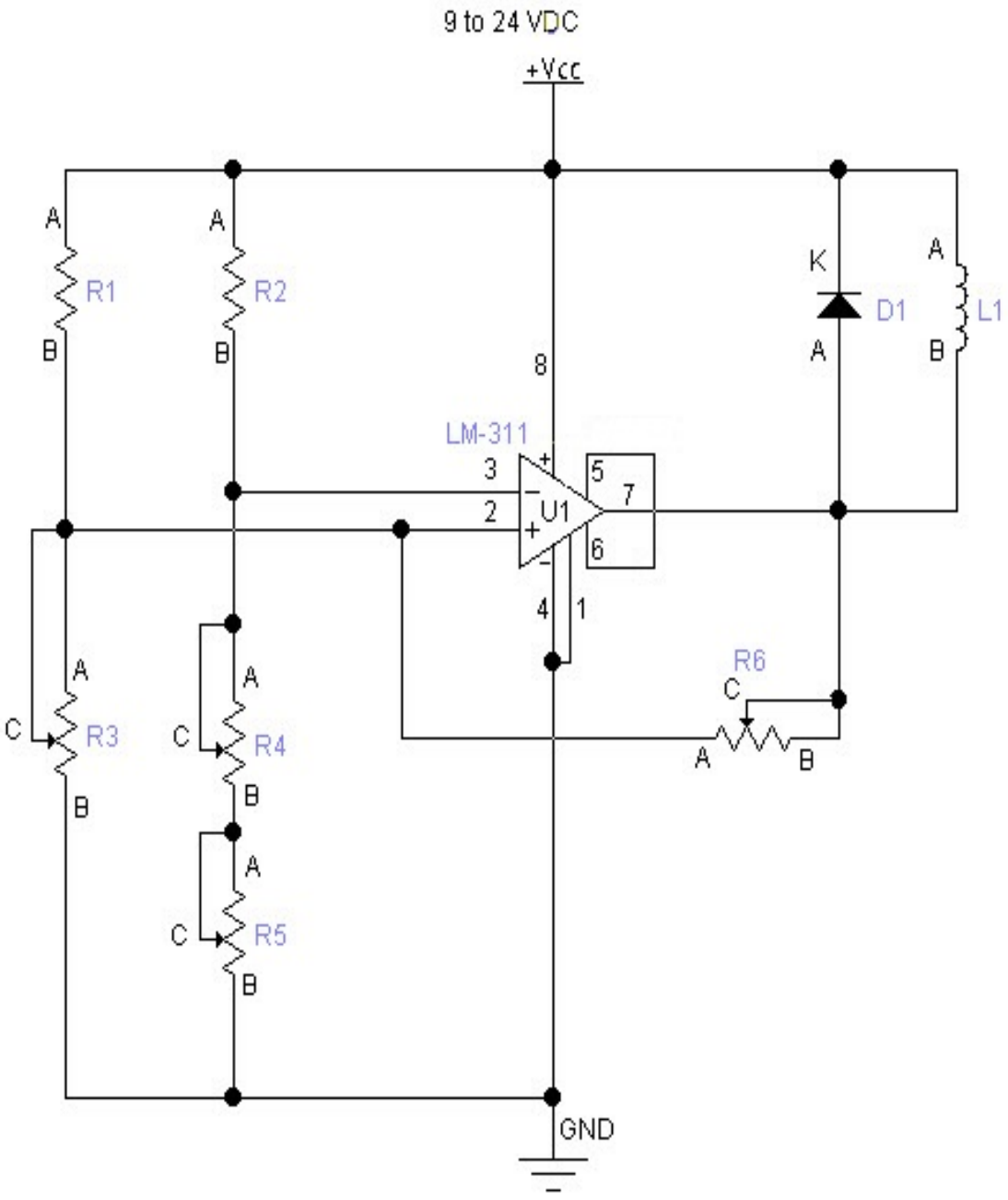
Circuit _____				**Node List**	Name	_____
WIRE	Clr	COLOR (clr)	NODE	PRINT CLEARLY use commas between con	CON	LOCATION
CNT	smpl	NAME	#	CONSTITUENTS (CON) add side indicators	{ # }	on schematic
swatches				example R-1 A, C-3+, D-2 Red -,		
					{ }	

Simple Starter Circuit DRAWING 1 Name: _____



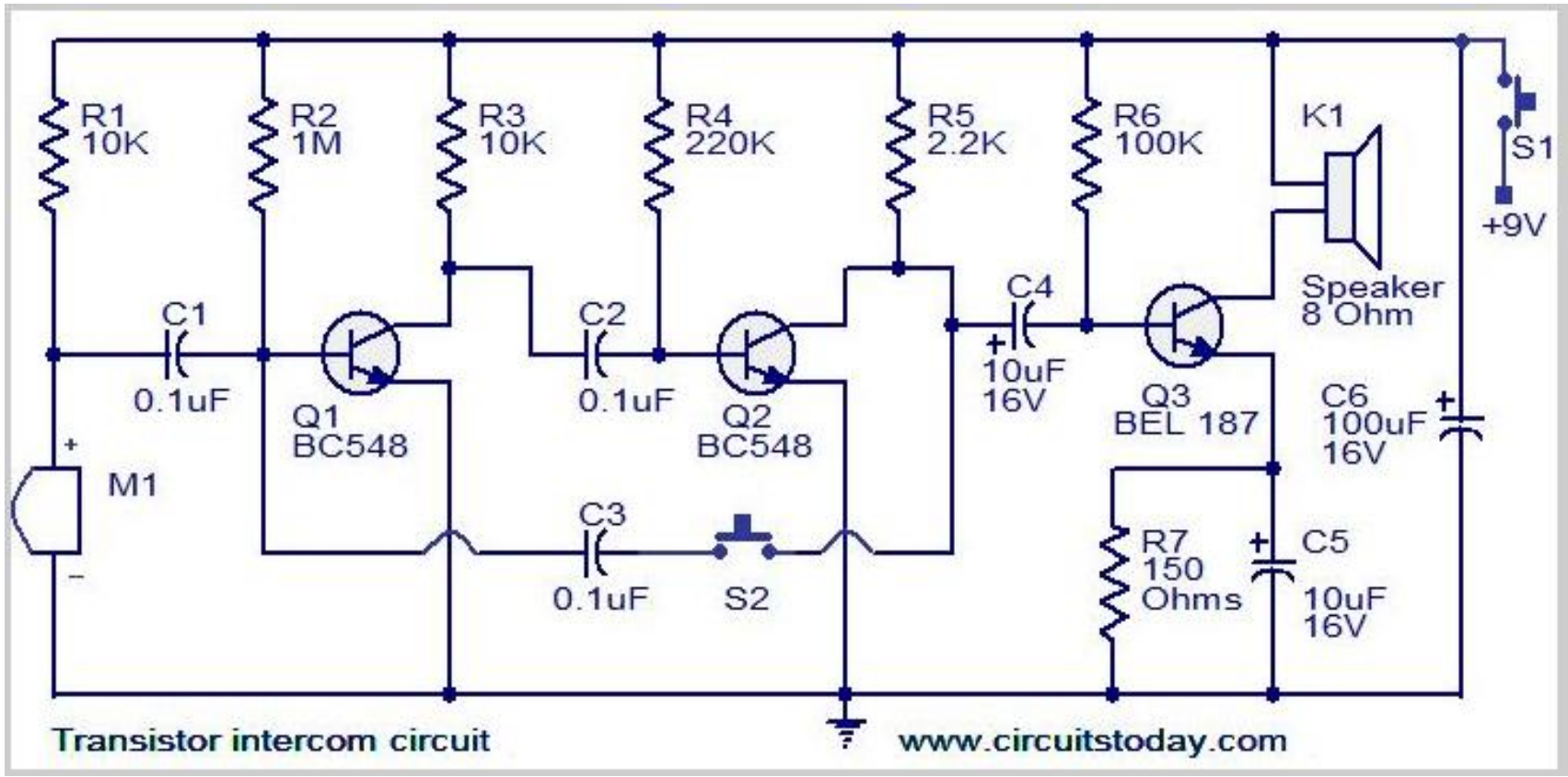
Circuit _____				**Node List**	Name	_____
WIRE	Clr	COLOR (clr)	NODE	PRINT CLEARLY use commas between con	CON	LOCATION
CNT	smpl	NAME	#	CONSTITUENTS (CON) add side indicators	{ # }	on schematic
swatches				example R-1 A, C-3+, D-2 Red -,		
					{ }	

Voltage Comparator Schematic Drawing DRAWING 2 Name: _____



Circuit _____				**Node List**	Name	_____
WIRE	Clr	COLOR (clr)	NODE	PRINT CLEARLY use commas between con	CON	LOCATION
CNT	smpl	NAME	#	CONSTITUENTS (CON) add side indicators	{ # }	on schematic
swatches				example R-1 A, C-3+, D-2 Red -,		
					{ }	

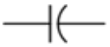
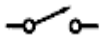
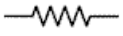
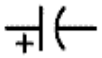
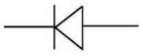
Transistor intercom Schematic Drawing 3 Drawing 3 Name: _____



PRE / POST **Basic Electronic Symbol Quiz** **Name:_____**

First, write your name on the quiz. Second, circle whether this is a pre or post quiz.

The object of this quiz is to see if the learner can identify the common electronic symbols, used in drafting and printed circuit board design layouts. Match the component symbol with the component name/number, by filling-in-the-blank (right column) with the number found next to that name in the center of the page. Then, you are to fill-in-the-blank (left column), with the same number, for the appropriate common letter designation used in industry for that component. The components symbols on the quiz are the same type that will be found on the schematic drawings, which will be used in the node layout. The letter(s) designators are those that will be found on the schematic drawings and used on the node-listing form.

GND	____	1 - Battery (multi cell)	____	
J	____	2 - Capacitor (mica, stack, non-polarized)	____	
K	____	3 - Capacitor polarized	____	
Q	____	4 - Diode	____	
R	____	5 - Ground	____	
C	____	6 - Jack	____	
R	____	7 - Plug	____	
SW	____	8 - Microphone	____	
P	____	9 - Potentiometer	____	
R	____	10 - Push Switch	____	
B	____	11 - Resistor	____	
S	____	12 - Speaker	____	
D	____	13 - Switch (SPST)	____	
C	____	14 - Thermistor	____	
M	____	15 - Transistor (NPN)	____	

Survey of Node layout and Node-listing

1 - What did you like best in this testing project?

2 - -What did you like least in this testing project?

3 - What did you learn specifically?

4 - What was the hardest part for you?

5 - What was the easiest part for you?

6 - What improvements, please be honest, would you make in the materials, PowerPoints, instructions, etc.?

Please take this evaluation on the Node-layout / Node-listing project.

Please **circle** the most appropriate response to the statement, thank you.

1. -- The PowerPoint on Learning Basic Electronic Symbols was presented in a clear manner that facilitated learning.

Strongly Agree Agree Neutral Disagree Strongly Disagree

2. -- The PowerPoint on How to do a node-layout was presented in a clear manner that facilitated learning.

Strongly Agree Agree Neutral Disagree Strongly Disagree

3. -- The additional instructional materials were helpful, clear and easy to use.

Strongly Agree Agree Neutral Disagree Strongly Disagree

4. -- The pre and post quiz directions were easy to understand and follow.

Strongly Agree Agree Neutral Disagree Strongly Disagree

5. -- The node-layout directions were easy to understand and follow.

Strongly Agree Agree Neutral Disagree Strongly Disagree

6. -- The node-listing directions were easy to understand and follow.

Strongly Agree Agree Neutral Disagree Strongly Disagree

7. -- The Simple Starter Circuit schematic drawing was easy to node-layout and node-list

Strongly Agree Agree Neutral Disagree Strongly Disagree

8. -- The Voltage Comparator schematic drawing was easy to node-layout and node-list

Strongly Agree Agree Neutral Disagree Strongly Disagree

9. -- The Transistor Intercom schematic drawing was easy to node-layout and node-list

Strongly Agree Agree Neutral Disagree Strongly Disagree

Please add any comments to help improve the instruction, and the materials associated with this project. Thank you for your participation in improving this class.

Interview Questions:

What was the easiest part of this project to understand? Please explain

What was the hardest part of this project to understand? Please explain

Did the instructional materials (PowerPoints and Handouts) help explain in detail what was expected and make it more understandable?

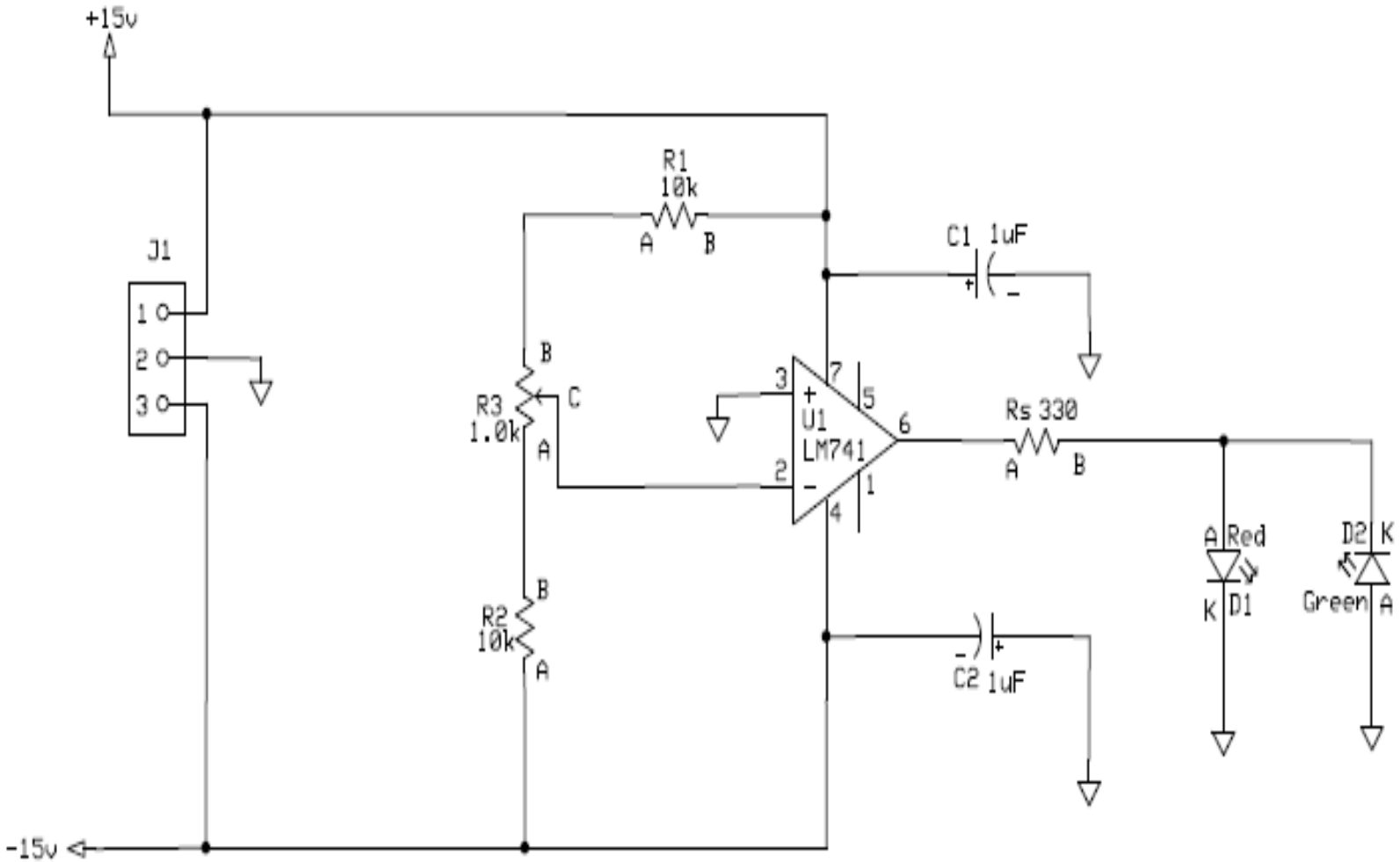
Was the node-layout / node-listing example clear and helpful as to what is required?

Should this project be added to the regular curriculum?

Was this material organized in a manner that it made it easy to follow? Please explain

Do you think you will use this information in the future? Will you use this knowledge in your chosen profession?

Circuit _____				**Node List**	Name	_____
WIRE	Clr	COLOR (clr)	NODE	PRINT CLEARLY use commas between con	CON	LOCATION
CNT	smpl	NAME	#	CONSTITUENTS (CON) add side indicators	{ # }	on schematic
swatches				example R-1 A, C-3+, D-2 Red -,		
					{ }	


+15v
-15v
J1
1
2
3
R1
10k
A
B
C1 1uF
+
-
R3
1.0k
B
C
A
R2
10k
B
A
U1
LM741
3
2
7
5
6
4
1
Rs 330
A
B
A Red
K D1
D2 K
Green A
C2 1uF
-
+