Assignment 2

Schoelen EET122, Spring 2020 Remote

RevB: 04/09/2020

Contents

Week2: Start 04/09/20 - End 04/16/20

Objectives:

1. Gain familiarity with Ki Cad schematic capture by doing a few simple KiCad tasks
2. Continue to gain knowledge of circuits by building and debugging.
3. Exposure to synchronous circuits (flip-flops, yeah!)
4. Gain familiarity with Arduino IDE by performing a simple task.

Read / Watch:

Last week's links if you need a review.

Do:

Use the Lab2 Discussion board if you need help.

See Appendix for additional details

1. Load into KICad: EET122_Lab2 from D2L
2. Complete the schematic using week1 schematic as a reference
    a. Add the 74ls74
    b. Add the two leds and two resistors
    c. Set the value of the resistors to 1k
    d. Annotate the design so that all components have a reference designator.
3. Complete the truth table in section A3 of the schematic
4. Do a sketch of your breadboard layout before you build
5. Be sure your UNO has the sketch loaded form Lab1
6. Build the circuit on a breadboard and connect to the Arduino as shown in the schematic (Based on the truth table derived in item #3 you should be able know if your circuit is operating correctly).
7. Once you have the circuit working, open the Arduino IDE and load sketch from week. At the top of the program is a #define that sets the clock time high and time low (in mili-seconds). Change the #define to 50. Program it into the UNO.

Deliverables to be uploaded to D2L for grading

For the deliverable use this this naming convention please: LastNameFirstName_L2

1. Upload the KiCad files, both the .pro and the .sch

2. Shoot a video of your circuit with LEDs blinking at the rate dictated by the new #define we did in item #7 above. Upload to d2l deliverables for lab2.

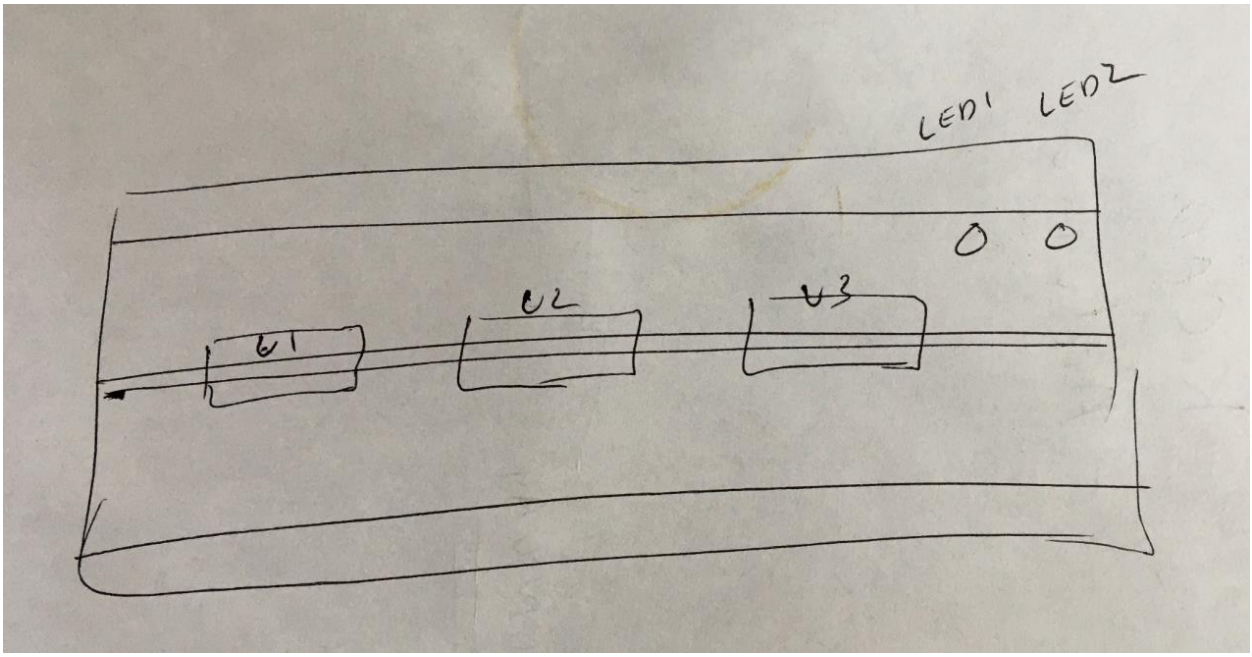
3. Scan or shoot a picture of you pre-layout sketch (item #4 above)

4. Document a conclusion. The conclusion is free from, it's a reflection on the lab. There's no minimum length. What you write is completely up to you. Upload conclusion to D2L deliverables Lab2. Some topics you cover may be:

    a) What did you learn?
    b) What did you struggle with?
    c) What was too easy?
    d) Observations if any
    e) Comments if any
    f) How many hours did this lab take you complete (all the tasks)

Appendix

Breadboard Sketch

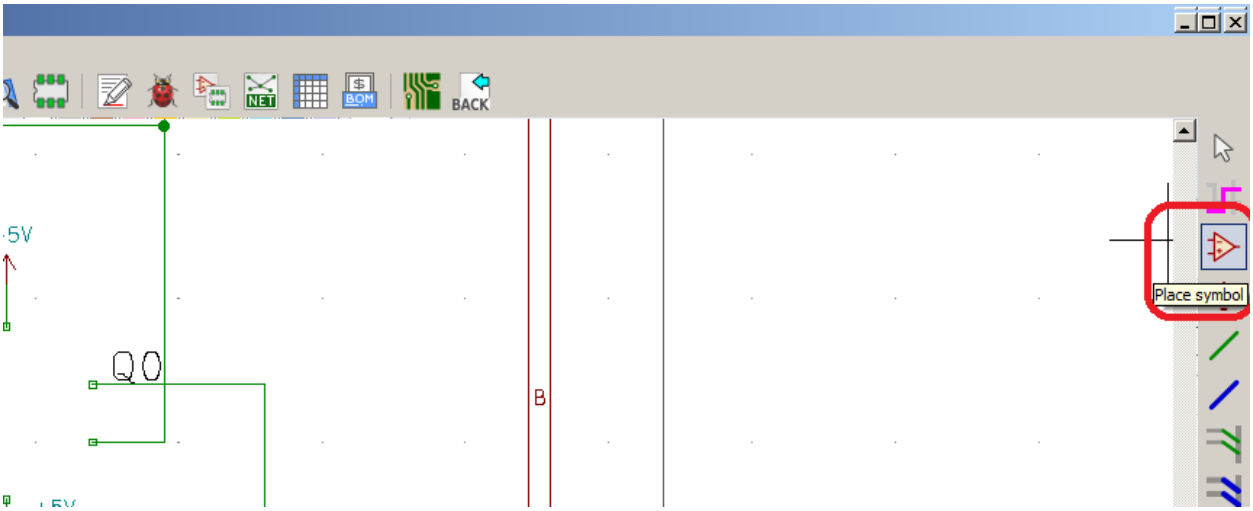


Kicad

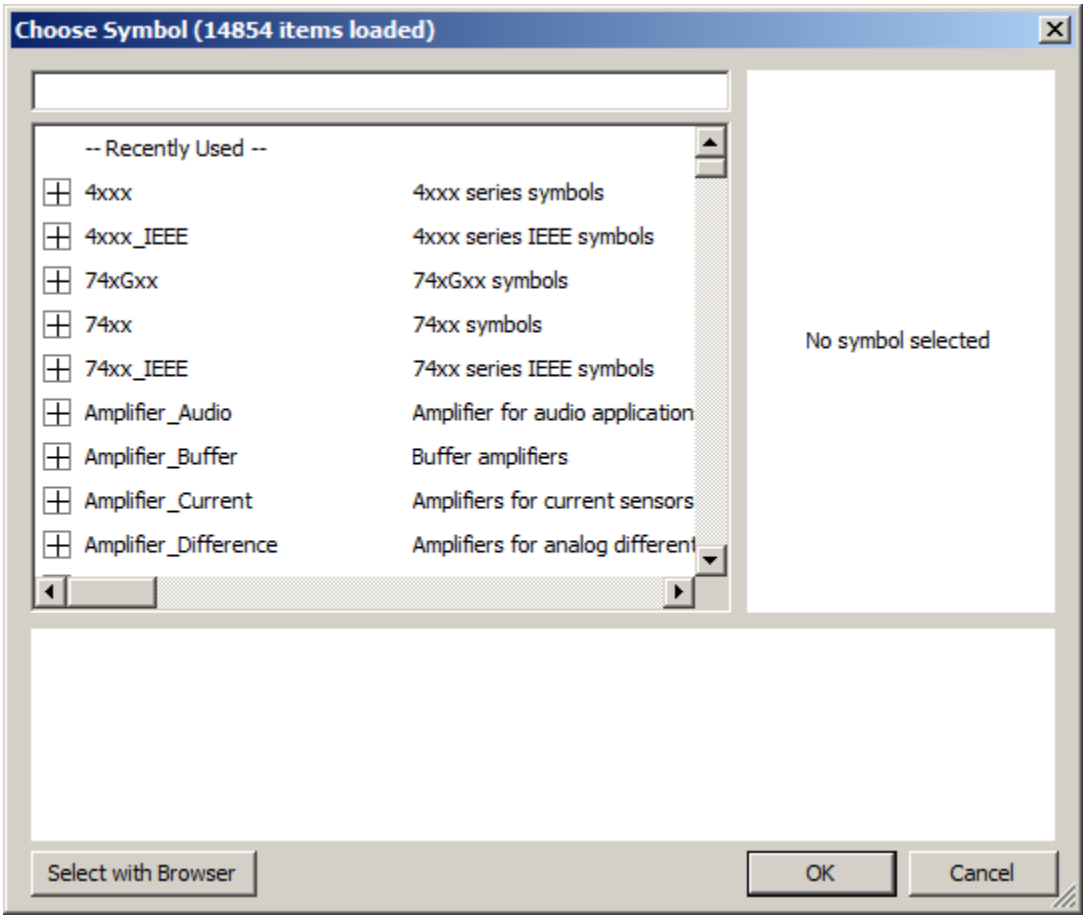
Place Component

To place a component:

1. Click the Place Symbol Icon



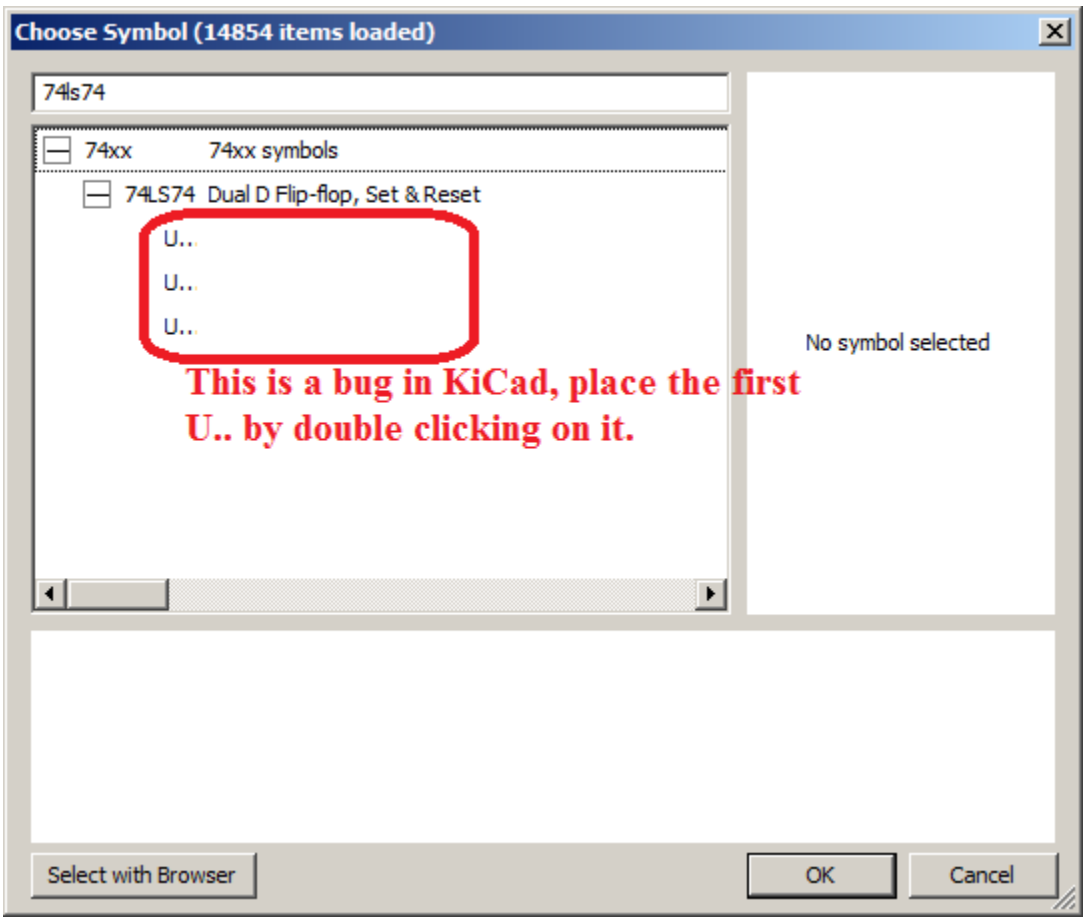
2. Click the workspace (a choose symbol box will open)



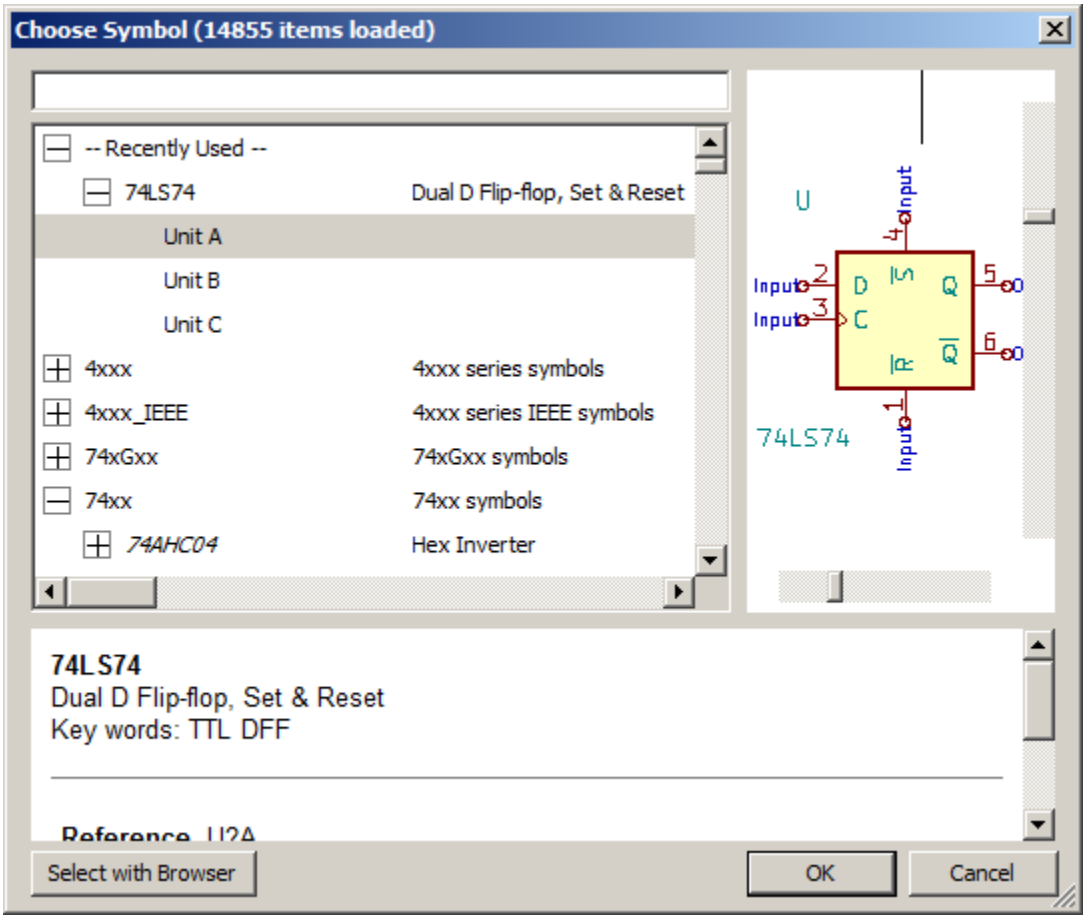
3. In filter type 74ls74

4. Expand the hierarchy to get to 74Ls74 Dual D Flip-Flip, Set & Reset

5. Double click on the first U.. and place the component in the schematic.

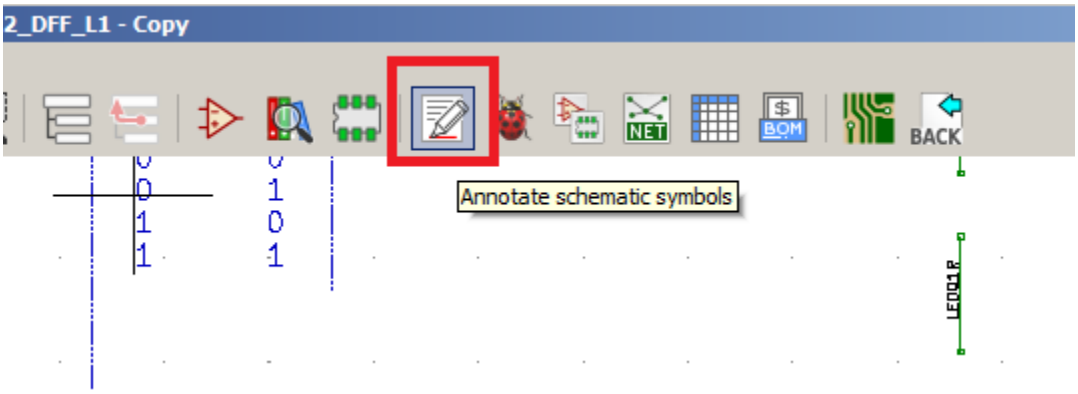


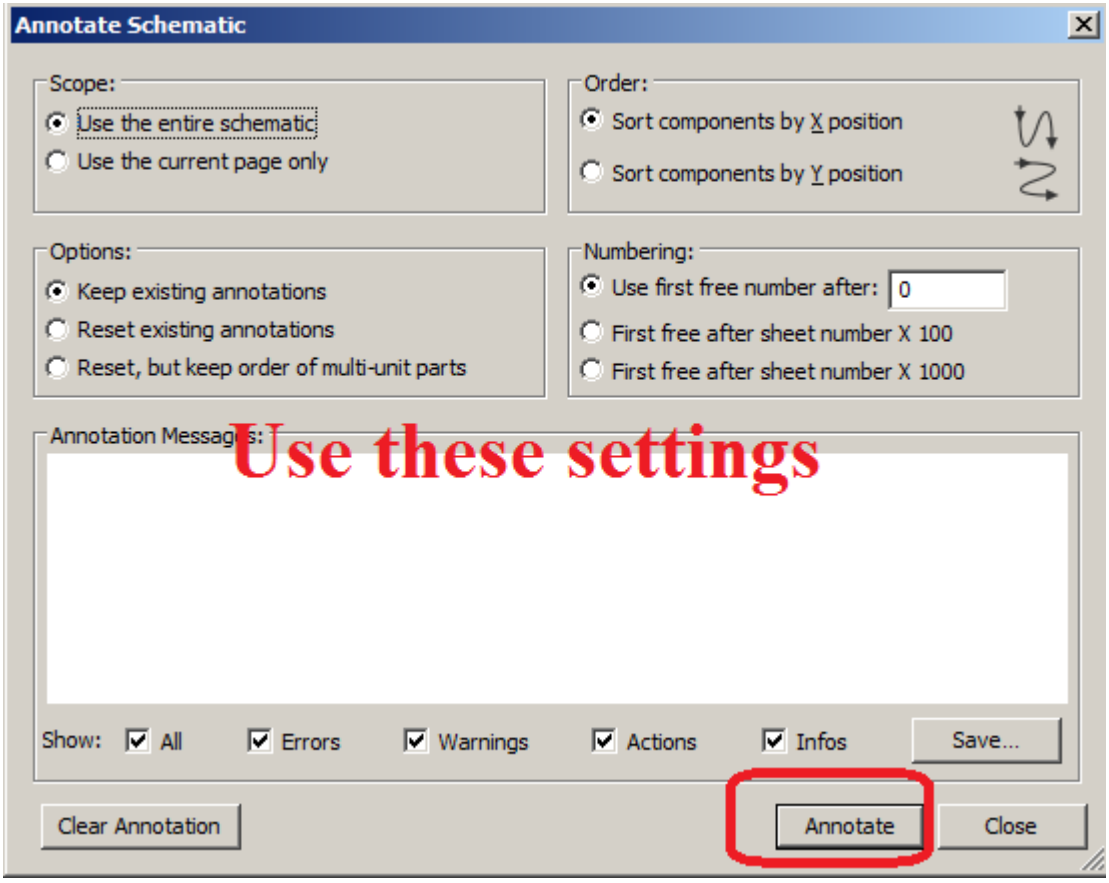
6. Place the remaining two components (Unit B and Unit C) of the 74LS74 one at a time.



Annotate

To annotate the design (that is, to automatically assign reference designators) click the Annotate schematic symbols icon.


2_DFF_L1 - Copy
NET
BOM
BACK
Annotate schematic symbols
LED01.R


Annotate Schematic
Scope:
Use the entire schematic
Use the current page only
Order:
Sort components by X position
Sort components by Y position
Options:
Keep existing annotations
Reset existing annotations
Reset, but keep order of multi-unit parts
Numbering:
Use first free number after: 0
First free after sheet number X 100
First free after sheet number X 1000
Annotation Messages:
Use these settings
Show: All Errors Warnings Actions Infos
Save...
Clear Annotation
Annotate
Close