

KiCad training

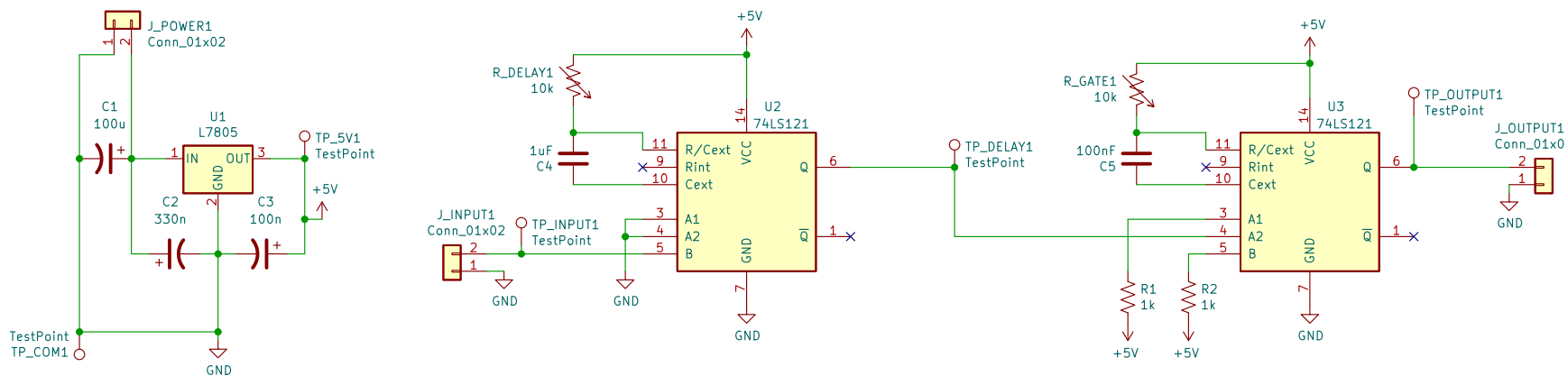
Blaise Thompson

March 21, 2022

Contents

Part of the training materials prepared by the [Chemistry Shops](#) at UW–Madison.
Source code and all associated files can be found at [GitHub](#).
If you find any mistakes or feel that any information is missing, please [open an issue](#).

reference	symbol	footprint
C1	Device:C_Polarized_US	Capacitor_THT:CP_Radial_D7.5mm_P2.50mm
C2	Device:C_Polarized_US	Capacitor_THT:CP_Radial_D7.5mm_P2.50mm
C3	Device:C_Polarized_US	Capacitor_THT:CP_Radial_D7.5mm_P2.50mm
C4	Device:C	Capacitor_THT:CP_Radial_D7.5mm_P2.50mm
C4	Device:C	Capacitor_THT:CP_Radial_D7.5mm_P2.50mm
J_INPUT	Connector_Generic:Conn_01x02	Connector_Molex:Molex_KK-254_AE-6410-02A_1x02_P2.54mm_Vertical
J_OUTPUT	Connector_Generic:Conn_01x02	Connector_Molex:Molex_KK-254_AE-6410-02A_1x02_P2.54mm_Vertical
J_POWER	Connector_Generic:Conn_01x02	Connector_Molex:Molex_KK-254_AE-6410-02A_1x02_P2.54mm_Vertical
R1	Device:R_US	Resistor_THT:R_Axial_DIN0207_L6.3mm_D2.5mm_P7.62mm_Horizontal
R2	Device:R_US	Resistor_THT:R_Axial_DIN0207_L6.3mm_D2.5mm_P7.62mm_Horizontal
R_DELAY	Device:R_Variable_US	Connector_Molex:Molex_KK-254_AE-6410-02A_1x02_P2.54mm_Vertical
R_GATE	Device:R_Variable_US	Connector_Molex:Molex_KK-254_AE-6410-02A_1x02_P2.54mm_Vertical
TP_5V	Connector:TestPoint	TestPoint:TestPoint_Loop_D2.60mm_Drill1.6mm_Beaded
TP_COM	Connector:TestPoint	TestPoint:TestPoint_Loop_D2.60mm_Drill1.6mm_Beaded
TP_DELAY	Connector:TestPoint	TestPoint:TestPoint_Loop_D2.60mm_Drill1.6mm_Beaded
TP_INPUT	Connector:TestPoint	TestPoint:TestPoint_Loop_D2.60mm_Drill1.6mm_Beaded
TP_OUTPUT	Connector:TestPoint	TestPoint:TestPoint_Loop_D2.60mm_Drill1.6mm_Beaded
U1	Regulator_Linear:L7805	Package_TO_SOT_THT:TO-220-3_Vertical
U2	74xx:74LS121	Package_DIP:DIP-16_W7.62mm_LongPads
U3	74xx:74LS121	Package_DIP:DIP-16_W7.62mm_LongPads
mounting		MountingHole_3.2mm_M3



blaise.thompson@wisc.edu

Blaise Thompson

Instrument Shop

Department of Chemistry

UW-Madison

Sheet: /

File: delay-generator.kicad_sch

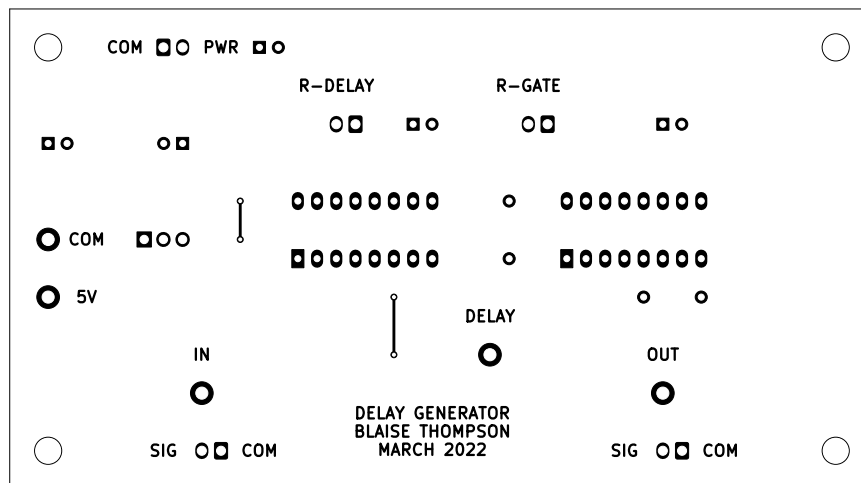
Title: Delay Generator

Size: USLetter Date: 2022-03-21

KiCad E.D.A. kicad 6.0.2+dfsg-1

Rev: A

Id: 1/1



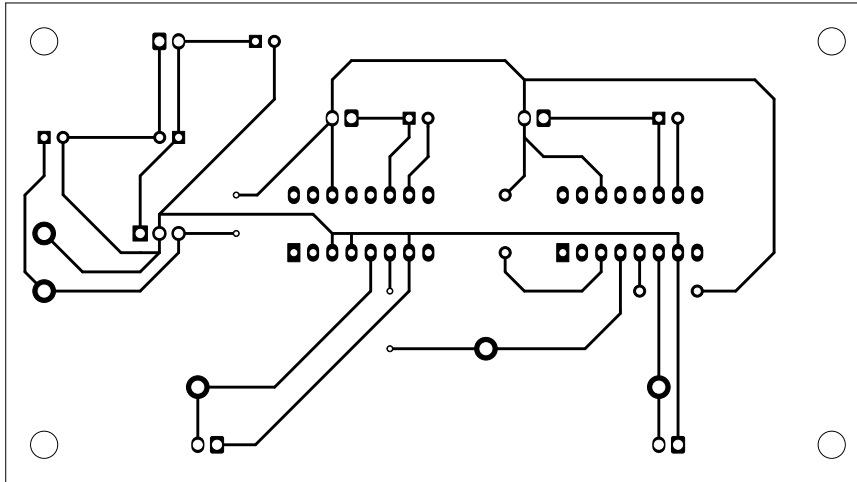
blaise.thompson@wisc.edu
Blaise Thompson
Instrument Shop
Department of Chemistry
UW-Madison

Sheet:
File: delay-generator.kicad_pcb

Title: Delay Generator

Size: USLetter Date: 2022-03-21
KiCad E.D.A. kicad 6.0.2+dfsg-1

Rev: A
Id: 1/1



blaise.thompson@wisc.edu Blaise Thompson Instrument Shop Department of Chemistry UW-Madison		
Sheet: File: delay-generator.kicad_pcb		
Title: Delay Generator		
Size: USLetter	Date: 2022-03-21	Rev: A
KiCad E.D.A. kicad 6.0.2+dfsg-1		Id: 1/1

1 electronic computer-aided design

Various options, including:

- EAGLE (part of autodesk family)
- ExpressPCB
- KiCAD
- Altium
- Cadence OrCAD

Prefer KiCad.

Most ECAD programs have two main interfaces to the circuit:

1. The schematic
2. The PCB

2 introduction to KiCad

download from <http://kicad-pcb.org/download/>

When you first launch KiCad, you will be met with a totally blank application.

KiCad is actually made up of several sub-programs. You will see these listed along the top of the page. They include:

- schematic layout editor
- symbol library editor
- PCB layout editor
- footprint library editor
- gerber viewer
- import bitmap
- calculator
- worksheet layout editor

For this introduction, we will be building a simple delay generator circuit from 628.

2.1 creating a project

Before we create our first project, a word about staying organized. A KiCad project is not a single file. Instead, a KiCad project is a *folder* with different files corresponding to different pieces of your project, like the schematic and (when appropriate) PCB. Furthermore, you will find yourself using and modifying *libraries* of electronic footprints and symbols—these will also become part of your project. It is imperative that the contents of this folder remain organized, or you risk “breaking” your project.

Use File New Project to create a new project. Choose the location and name of your new project. You will find that KiCad creates a folder at your chosen location. That folder is immediately populated with three files:

- kicad_pcb
- pro
- sch

3 schematic capture using eeschema

Change page layout.

4 pcb layout using pcbnew

Change page layout.