Circuit Design on a Budget

Class 3: Creating Our Schematic

February 27, 2019

Charles J. Lord, PE
President, Consultant, Trainer
Blue Ridge Advanced Design and Automation

Presented by:

DesignNews

Blue Ridge Advanced Design and Automation
Asheville, North Carolina

CEC CONTINUING EDUCATION CENTER

Digi-Key ELECTRONICS

This Week's Agenda

2/25 Introduction to EDA and Our Tools

2/26 Starting at the Beginning

2/27 Creating Our Schematic

2/28 Creating Our PCB Layout

3/1 Our Finished Board

This Week's Agenda

2/25 Introduction to EDA and Our Tools

2/26 Starting at the Beginning

2/27 Creating Our Schematic

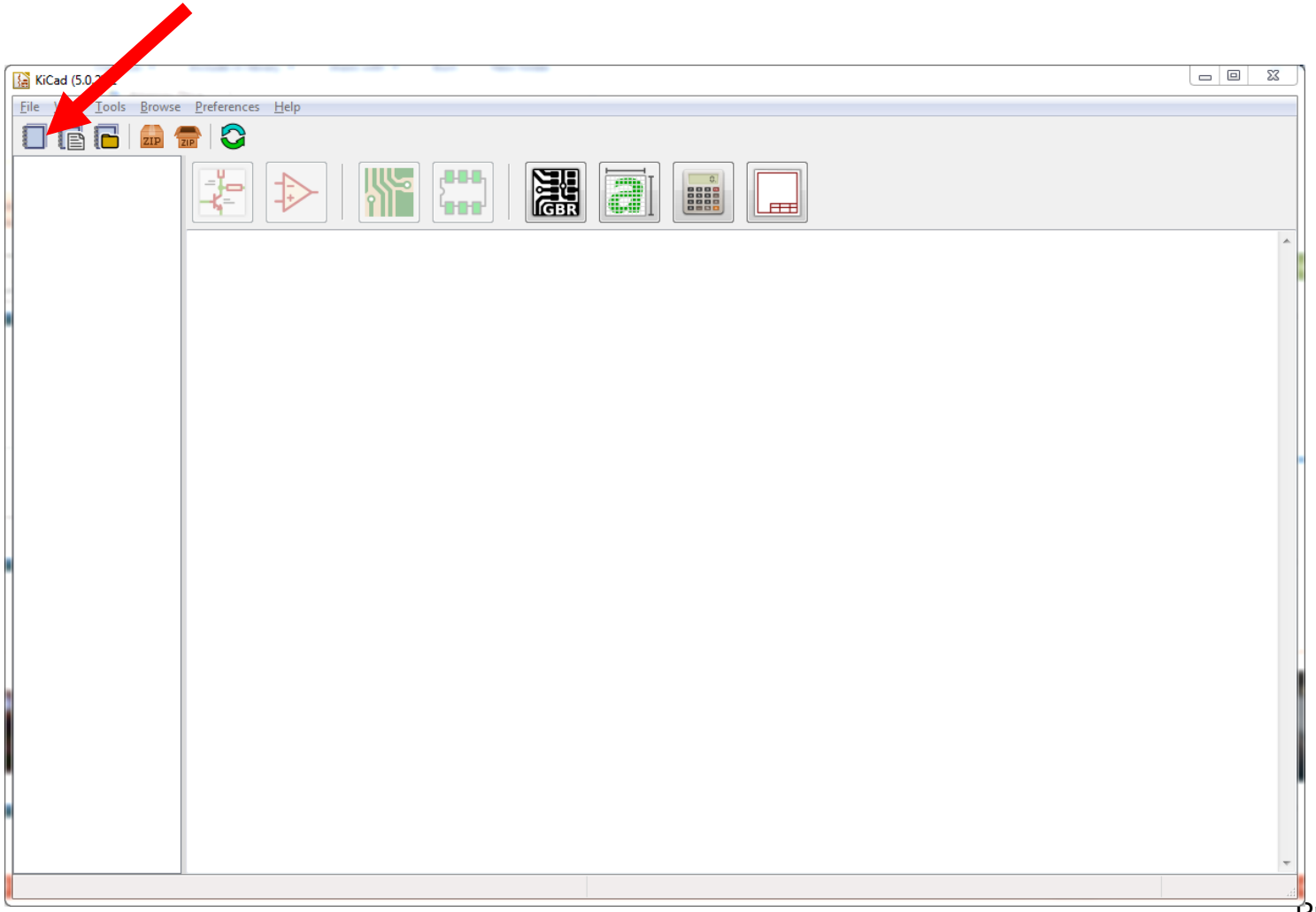
2/28 Creating Our PCB Layout

3/1 Our Finished Board

The EDA Two-Step

- I will be showing the remaining steps of creating our design and our board on both programs (CircuitStudio and KiCad), however we won't go over every step on each. I will emphasize differences when relevant
- Emphasis will be on KiCad
- The bottom line is to show each step generically as we go through our process.

Initializing - KiCad



Parts Sources - KiCad

- KiCad comes with a decent-sized library which is openly contributed to on Github.
- There is also a KiCad library provided by Digi-Key as well as other libraries on the web
- Important concept to remember in ANY EDA program – you should make your own local library to your circuit, with copies of library parts plus your own custom parts

Question 1 - Why create a local library?

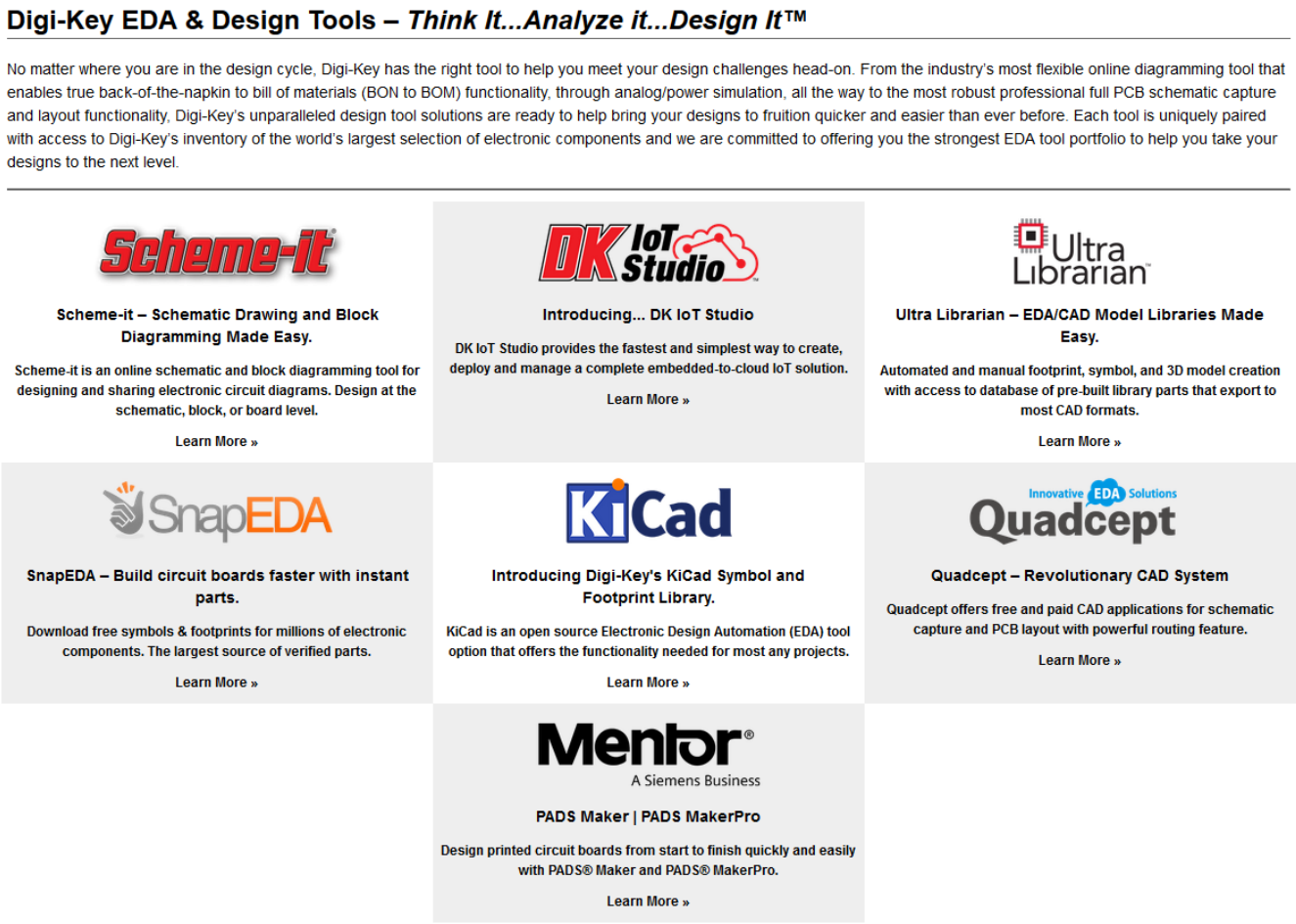
Parts Sources – CircuitStudio

- CircuitStudio comes with a very sparse library that seems circa 1990s… 2n3904 etc
- A subscription gets you access to the Altium Vault – which has some nice advantages (sharing parts) but is not as thorough (IMO) as KiCad – and has errors also
- Truism #2 – you will end up making parts

Example – our Microcontroller

- MKL25Z128VLH4 is complete part number
- 48 pin flat pack
- So where do we find it?

Digi-Key has Many Ties to EDA



D-K KiCad Page

Introducing Digi-Key's KiCad Symbol and Footprint Library and the KiCad EDA Tool

KiCad is an open source Electronic Design Automation (EDA) tool option that offers the functionality needed for almost any project. Because of its high level of functionality and no licensing fee, its popularity is exploding.

- KiCad is a full feature electronics development application for the design and manufacture of electronics that runs natively on Windows, OSX, and Linux.
- Application suite includes: schematic capture, printed circuit board layout, Gerber file viewer, solid model viewer, and much more.
- Python scripting support for board and footprint library automation.
- Large number of symbol, footprint, and model libraries included.
- Applications and documentation have been translated to multiple languages.

Digi-Key recognizes this trend and we are proud to offer a new KiCad library comprised of symbols and footprints created in-house by our own Digi-Key technicians and engineers.

- Over a thousand useful parts, with footprints associated(atomic)
- Built from the ground up by Digi-Key AEs
- Purpose built by hand and peer tested, no automation here
- Same open source license as KiCad Libraries
- Created with all parametric and ordering information

Our library is available on GitHub so you can link to our latest version.

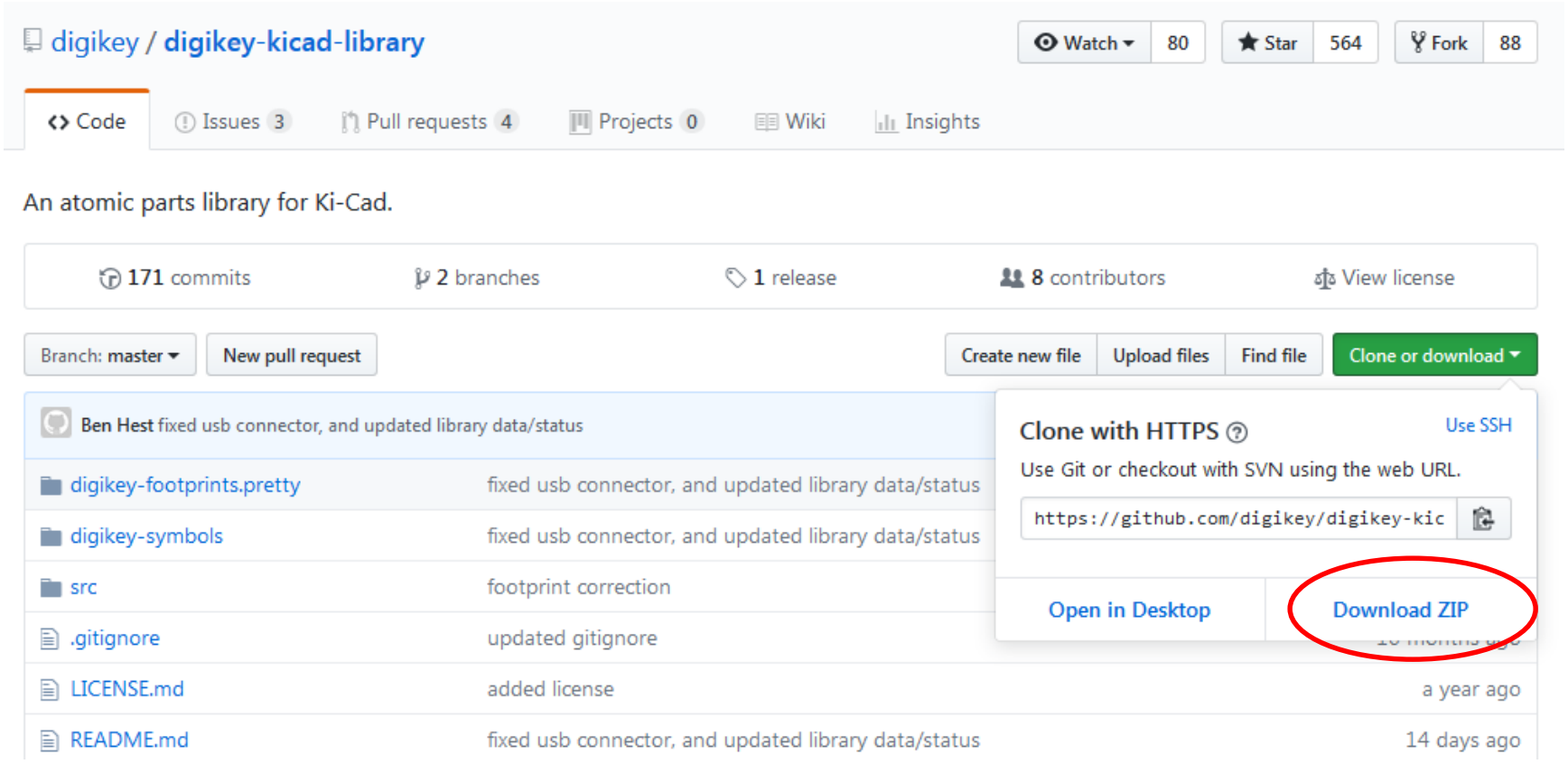
KiCad LIBRARY **Get it Now**

Are you new to KiCad? You can get the program here.

KiCad **Try it Now**

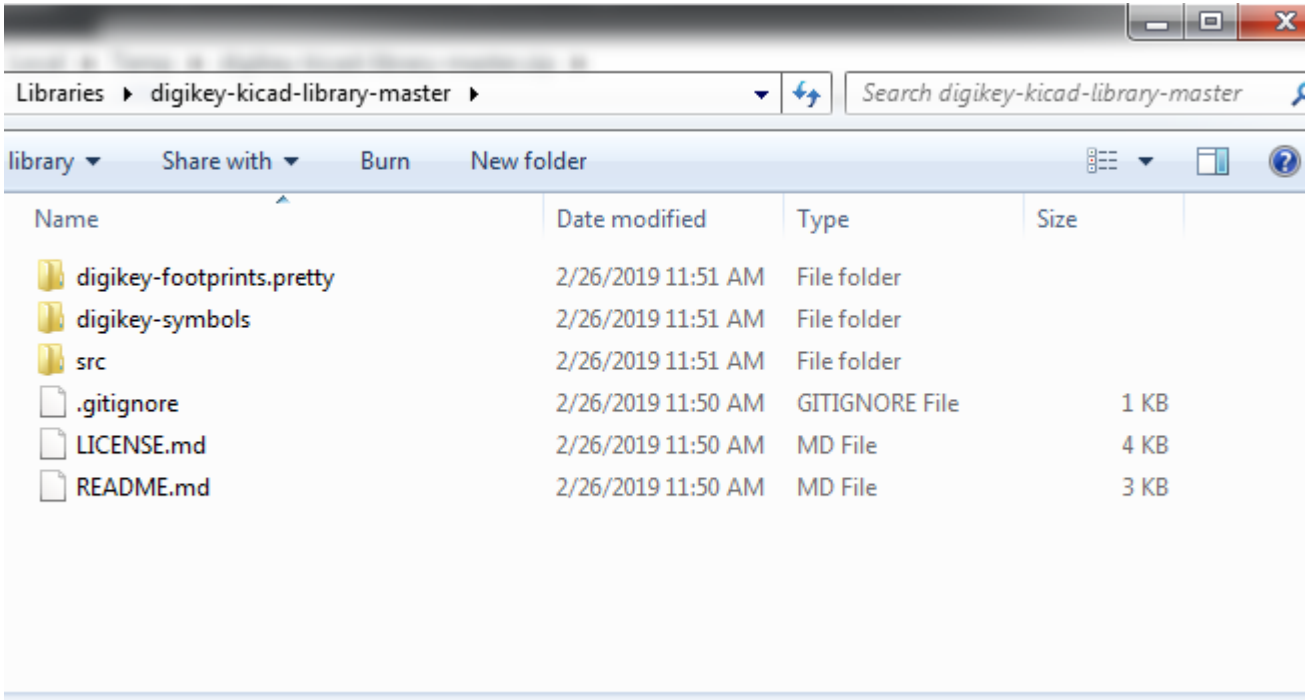
A copy of this library is also available here to Download Now.

Let’s get the Digi-Key Library!



Unzip to our Libraries Folder

- Users/public/kicad/libraries/

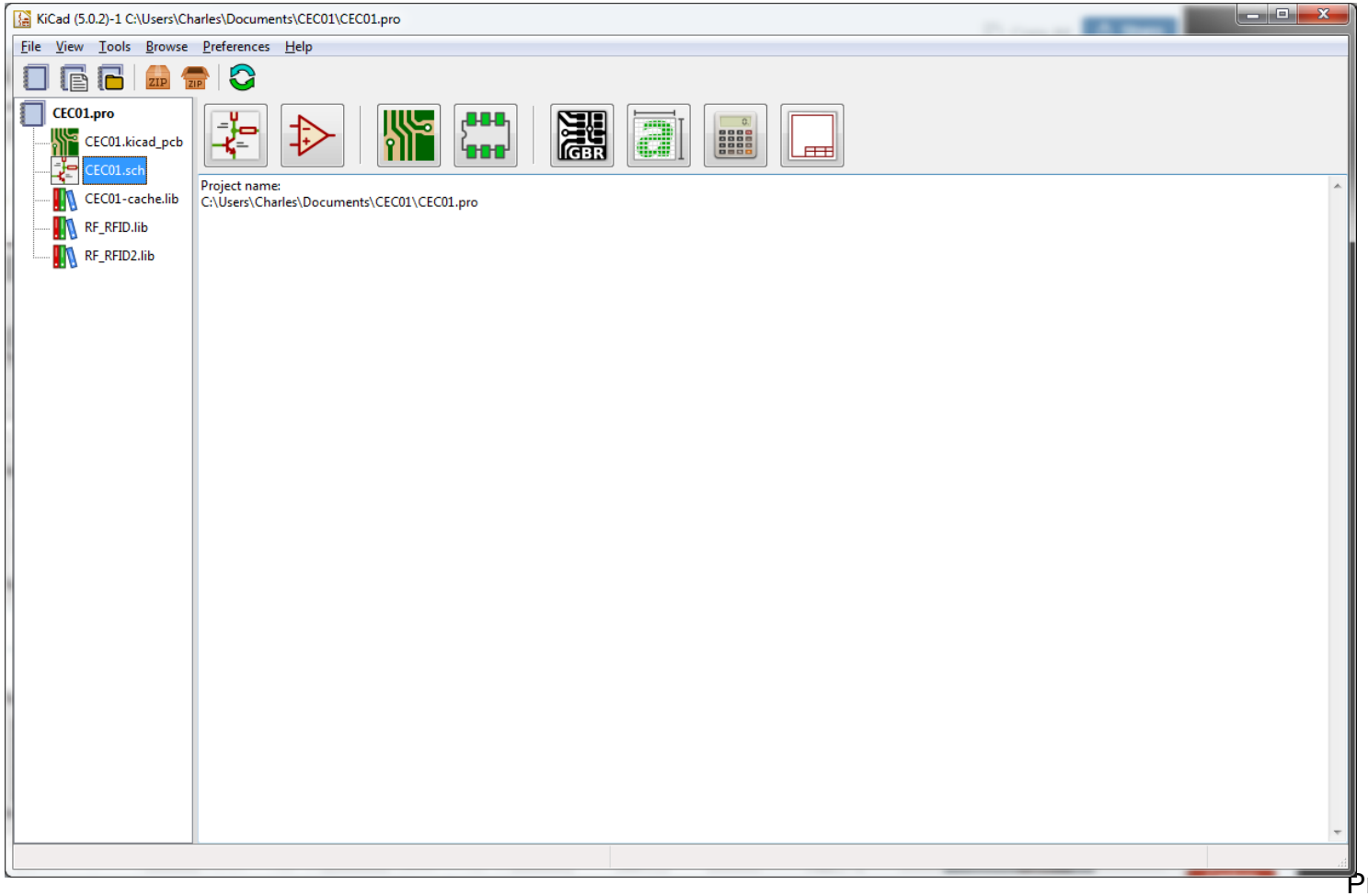


KiCad HOT KEYS

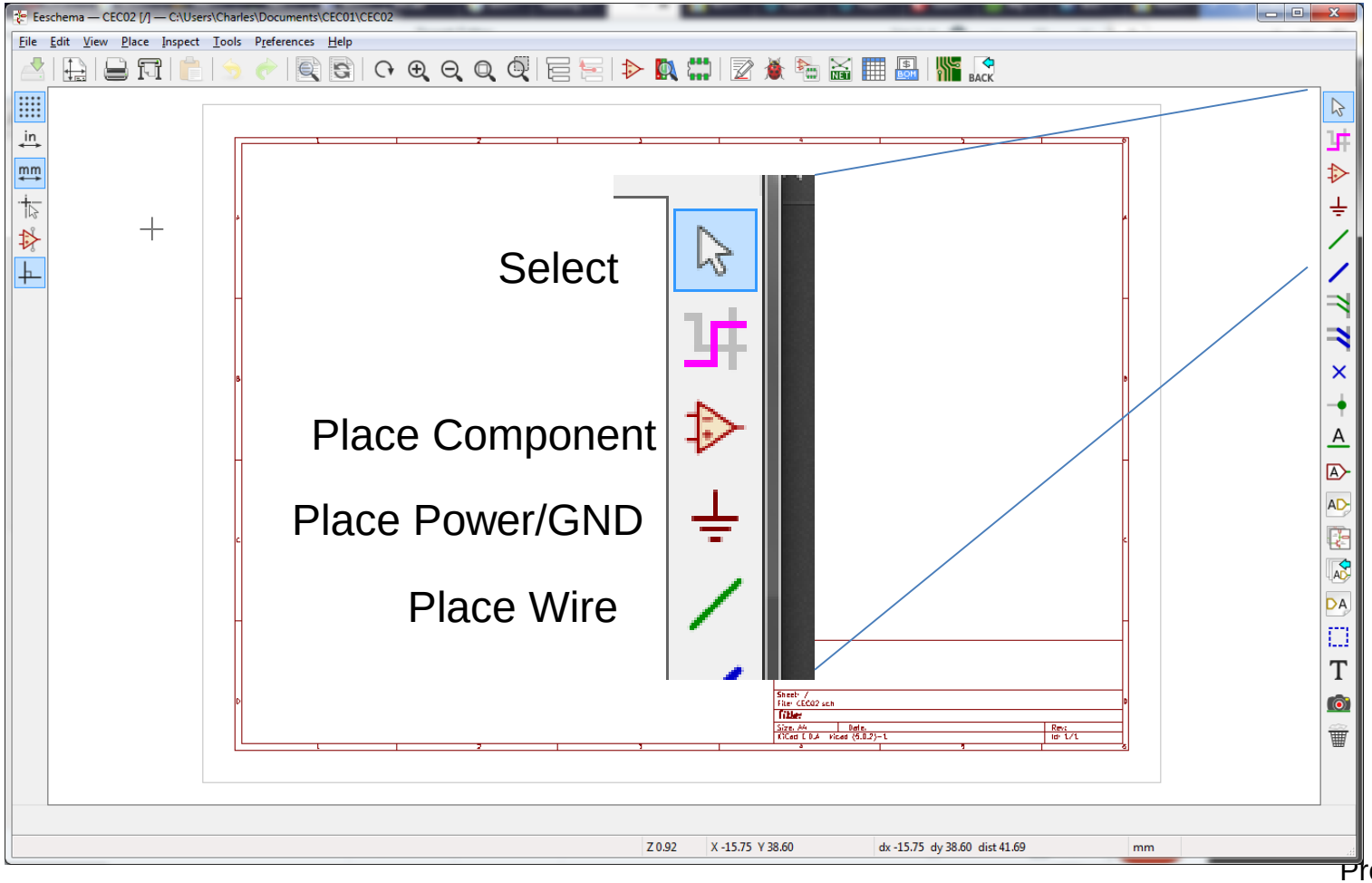
New	Ctrl+N
Open	Ctrl+O
Save	Ctrl+S
Save As	Ctrl+Shift+S
Print	Ctrl+P
Undo	Ctrl+Z
Redo	Ctrl+Y
Cut	Ctrl+X
Copy	Ctrl+C
Paste	Ctrl+V
Help (this window)	Ctrl+F1
Zoom In	F1
Zoom Out	F2
Zoom Redraw	F3
Zoom Center	F4
Fit on Screen	Home
Zoom to Selection	Ctrl+F5
Reset Local Coordinates	Space
Edit Item	E
Delete Item	Del
Rotate Item	R
Drag Item	G
Mouse Left Click	Return
Mouse Left Double Click	End
Find Item	Ctrl+F
Find Next Item	F5
Find Next DRC Marker	Shift+F5
Find and Replace	Ctrl+Alt+F
Repeat Last Item	Ins

Move Block → Drag Block	Tab
Move Schematic Item	M
Duplicate Symbol or Label	C
Add Symbol	A
Add Power	P
Mirror X	X
Mirror Y	Y
Orient Normal Component	N
Edit Symbol Value	V
Edit Symbol Reference	U
Edit Symbol Footprint	F
Edit with Symbol Editor	Ctrl+E
Begin Wire	W
Begin Bus	B
End Line Wire Bus	K
Add Label	L
Add Hierarchical Label	H
Add Global Label	Ctrl+H
Add Junction	J
Add No Connect Flag	Q
Add Sheet	S
Add Wire Entry	Z
Add Bus Entry	/
Add Graphic PolyLine	I
Add Graphic Text	T
Update PCB from Schematic	F8
Autoplace Fields	O
Leave Sheet	Alt+Back
Delete Node	Back
Highlight Connection	Ctrl+B

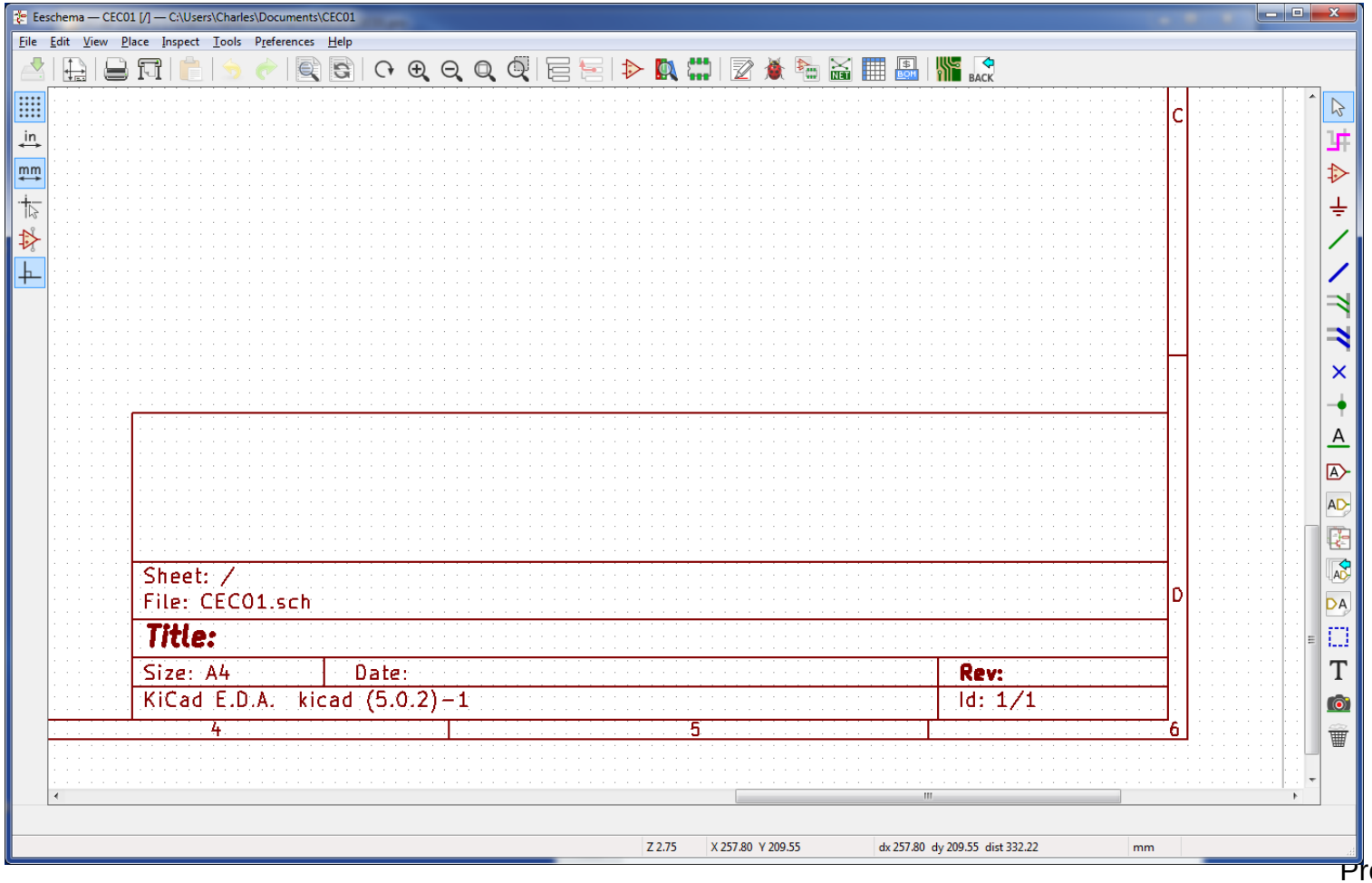
Click on the .sch file



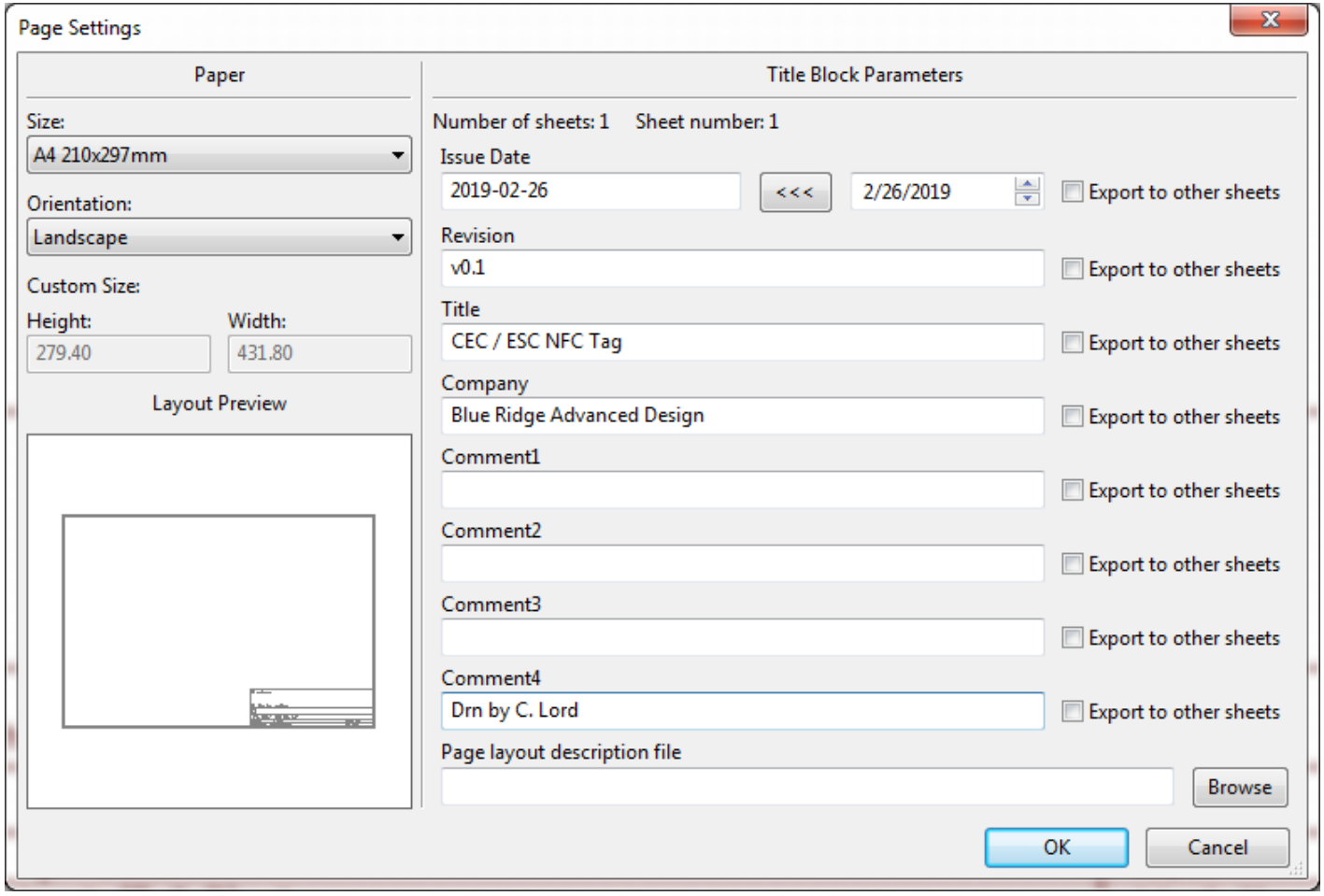
Let’s get started!



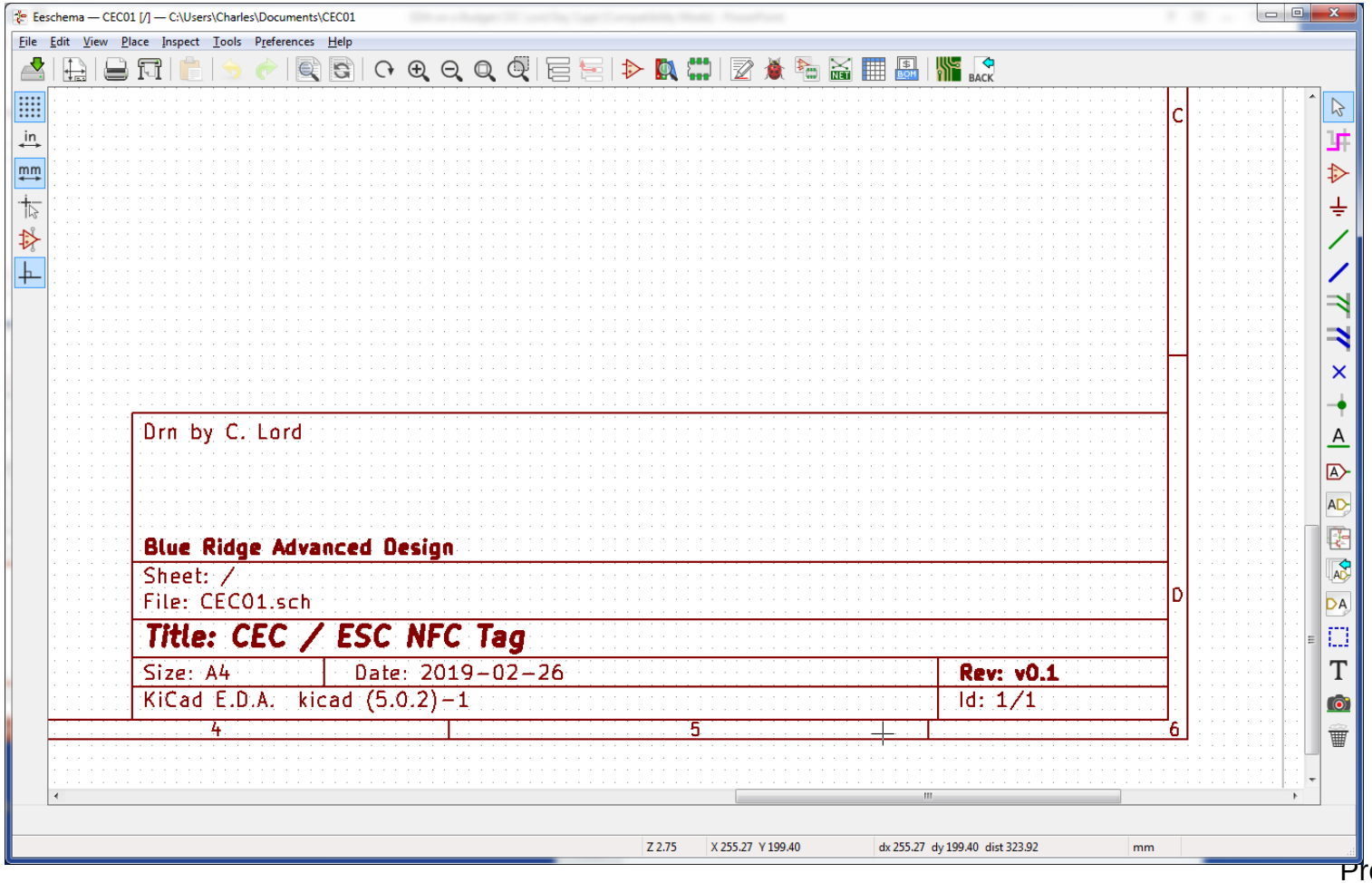
Title Block


Eeschema — CEC01 [/] — C:\Users\Charles\Documents\CEC01
File Edit View Place Inspect Tools Preferences Help
BACK
C
D
Sheet: /
File: CEC01.sch
Title:
Size: A4
Date:
Rev:
KiCad E.D.A. kicad (5.0.2)-1
Id: 1/1
4
5
6
Z 2.75
X 257.80 Y 209.55
dx 257.80 dy 209.55 dist 332.22
mm

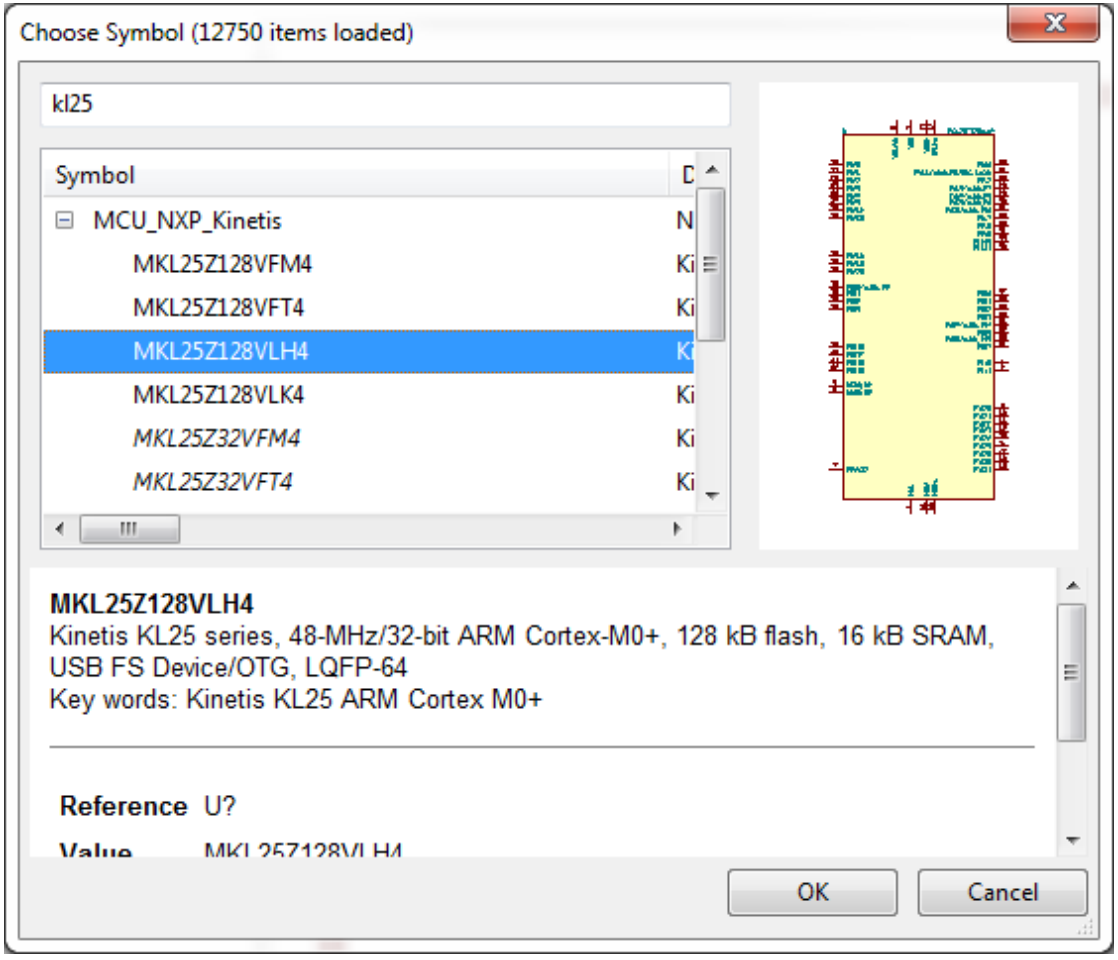
File, Page Settings



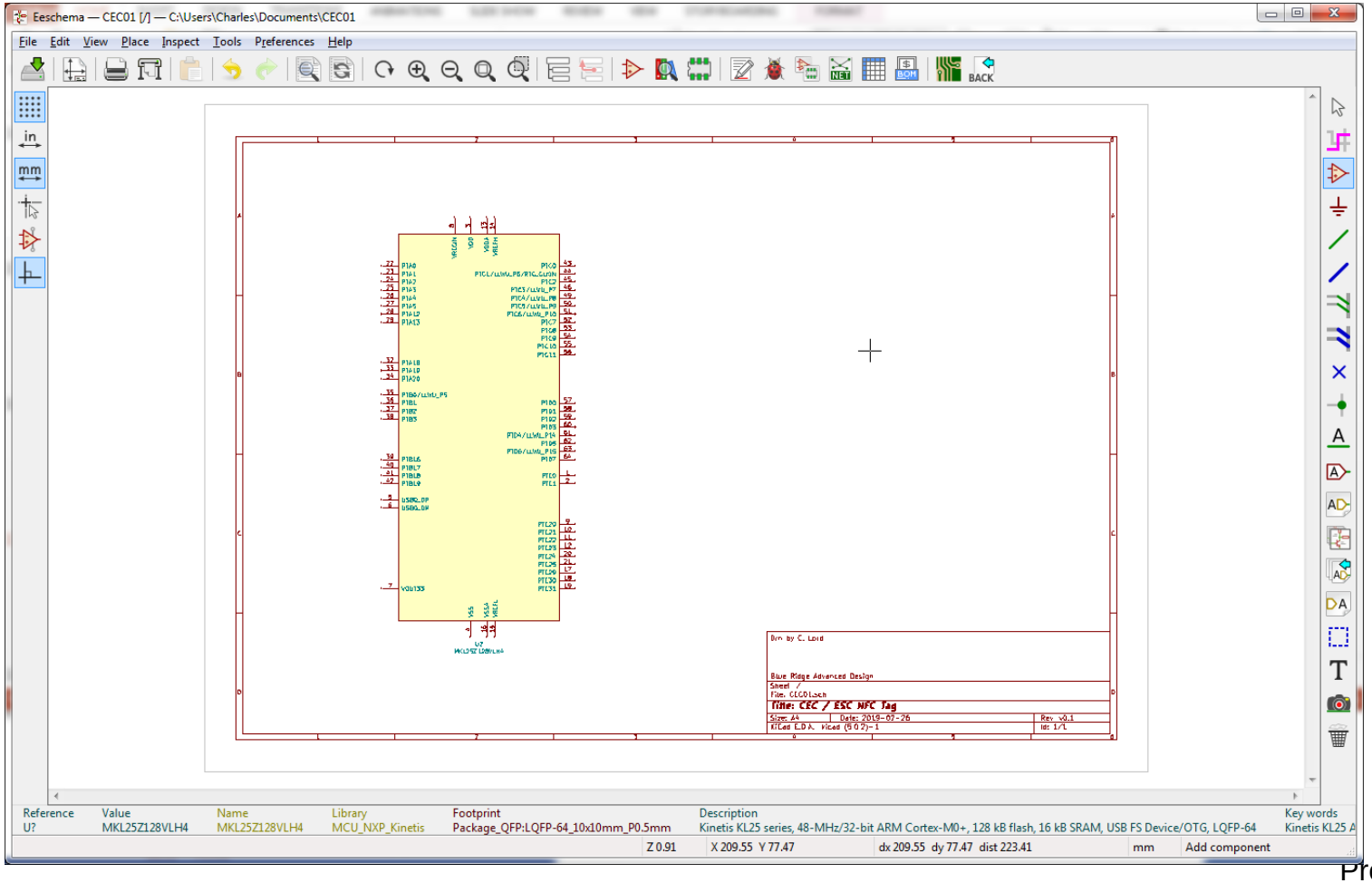
Comment 4 on Top



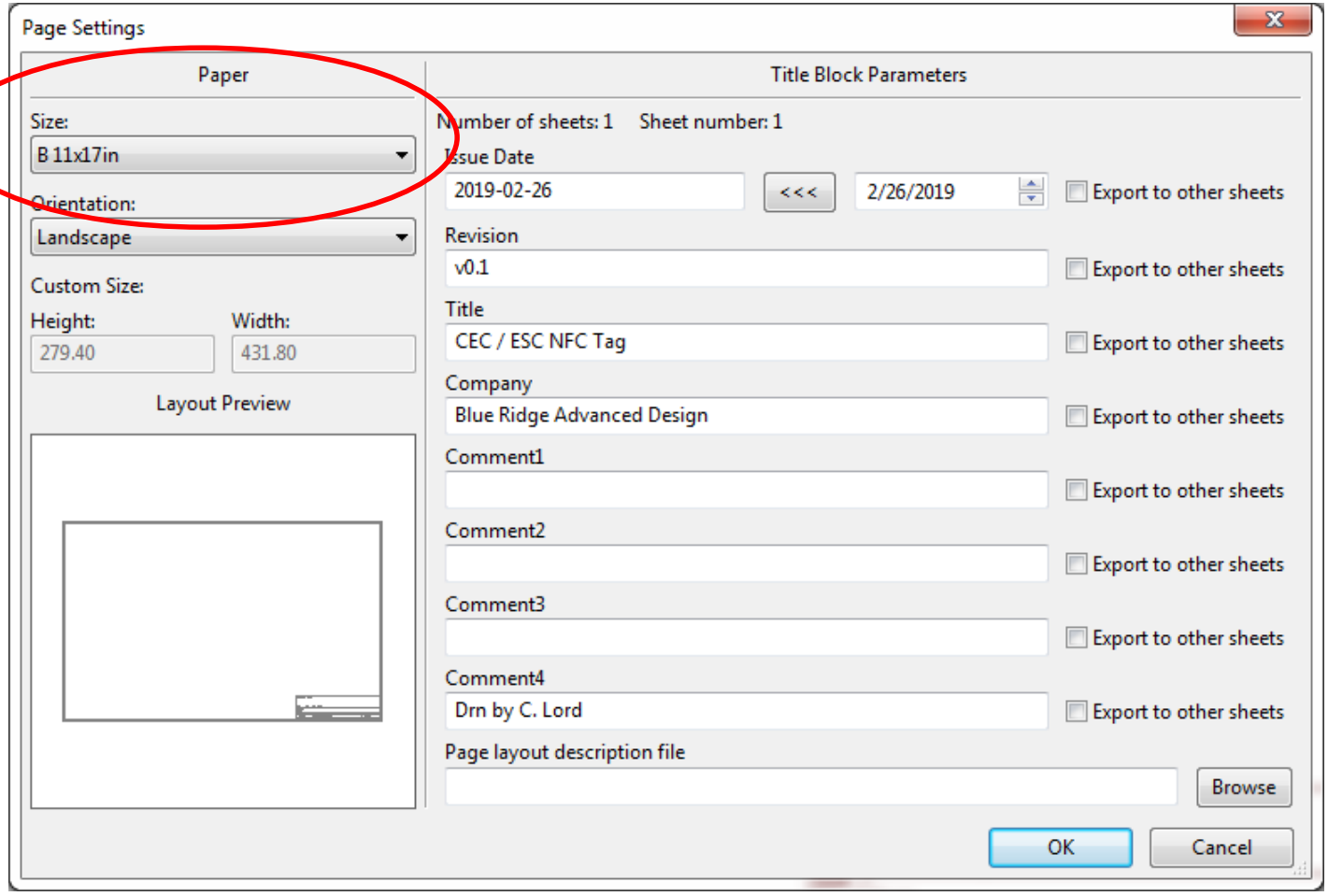
Built-in Library has our Chip



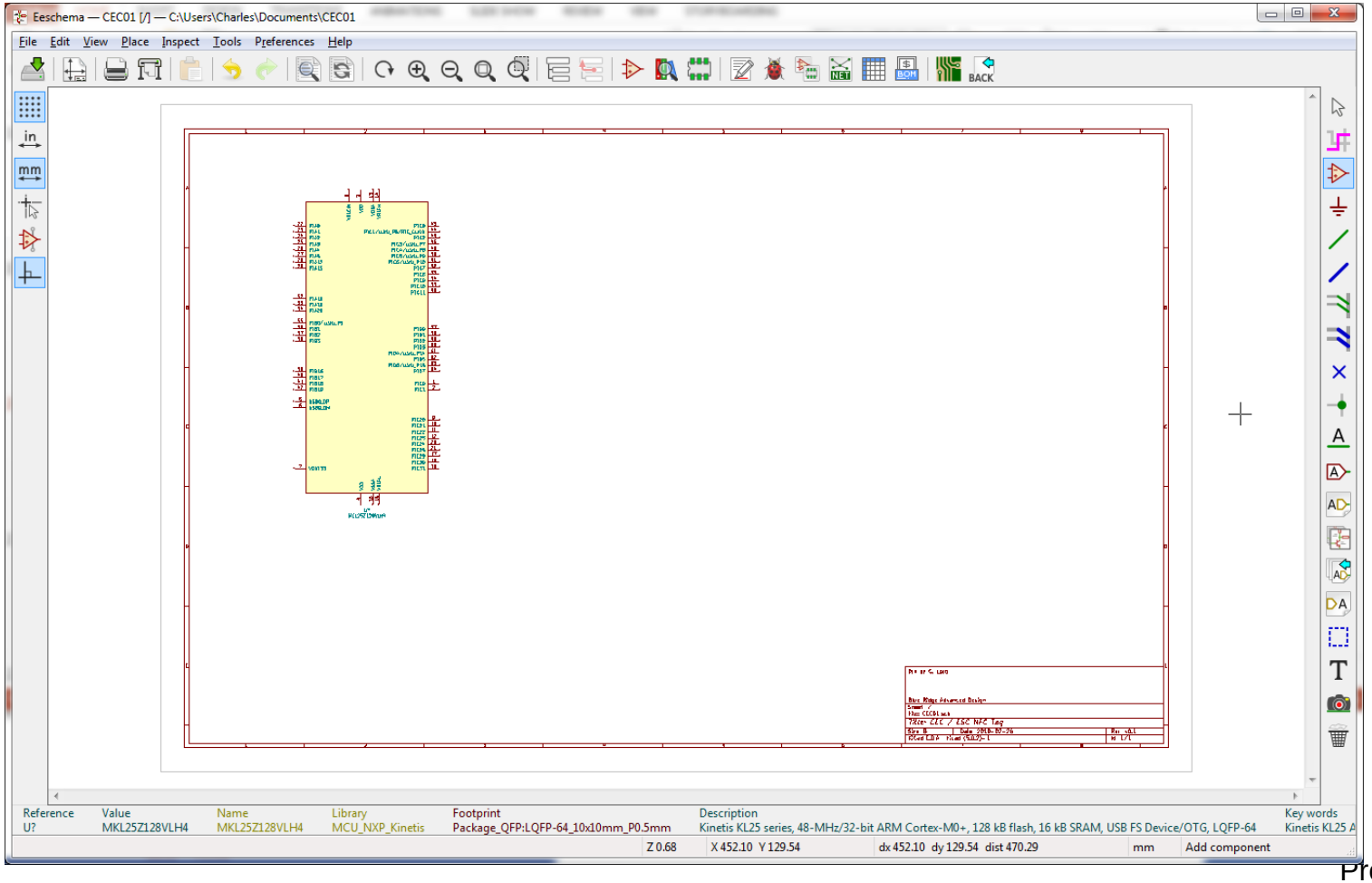
There is our Microcontroller



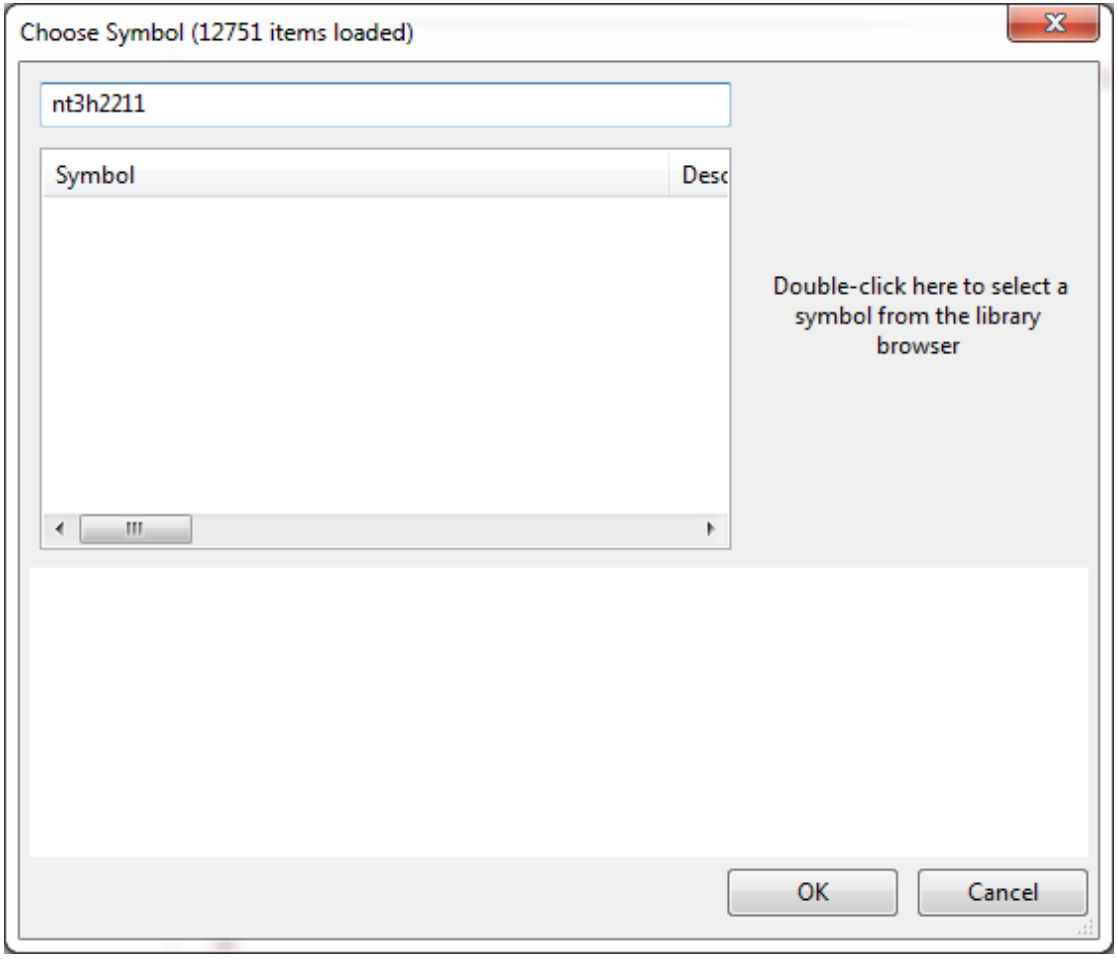
Change to B (11x17)



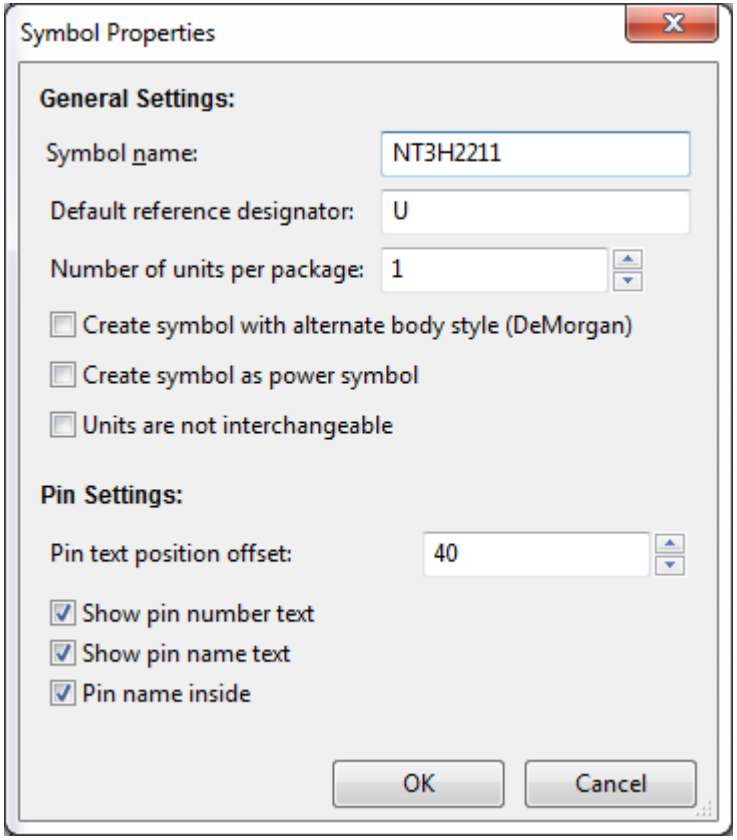
Now we have room



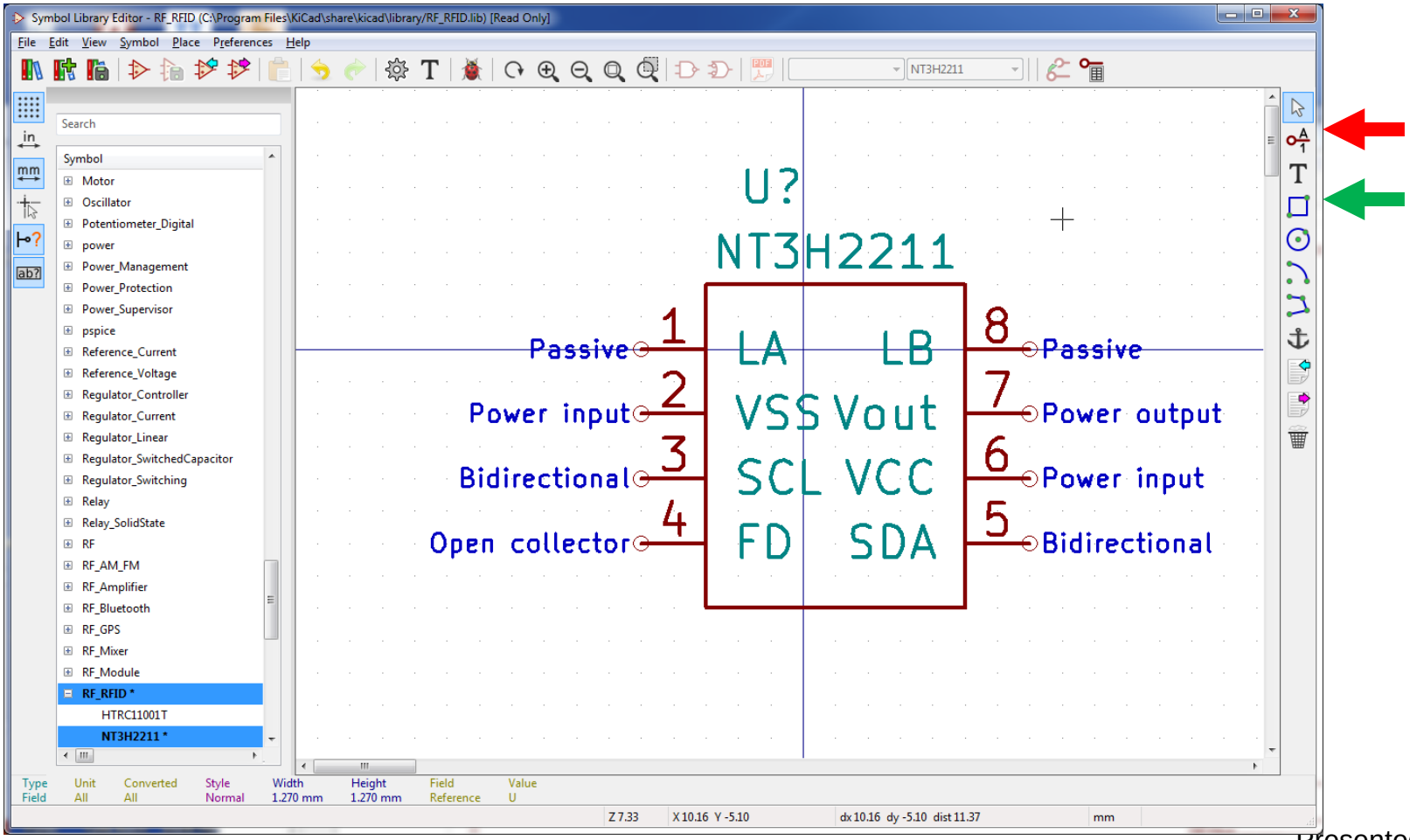
Now for the NTAG



Let's Create it!



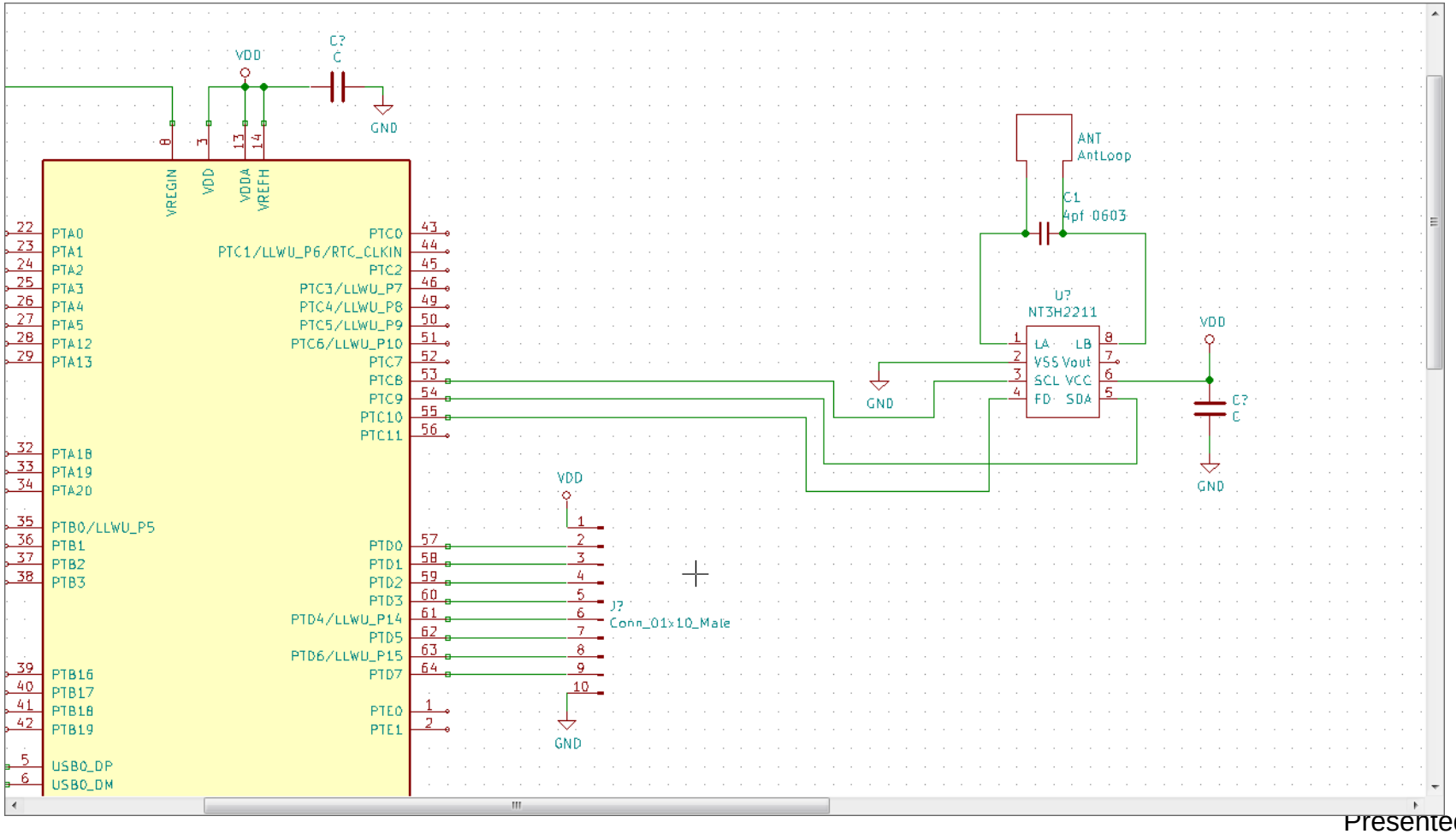
Define 8 pins, draw rectangle



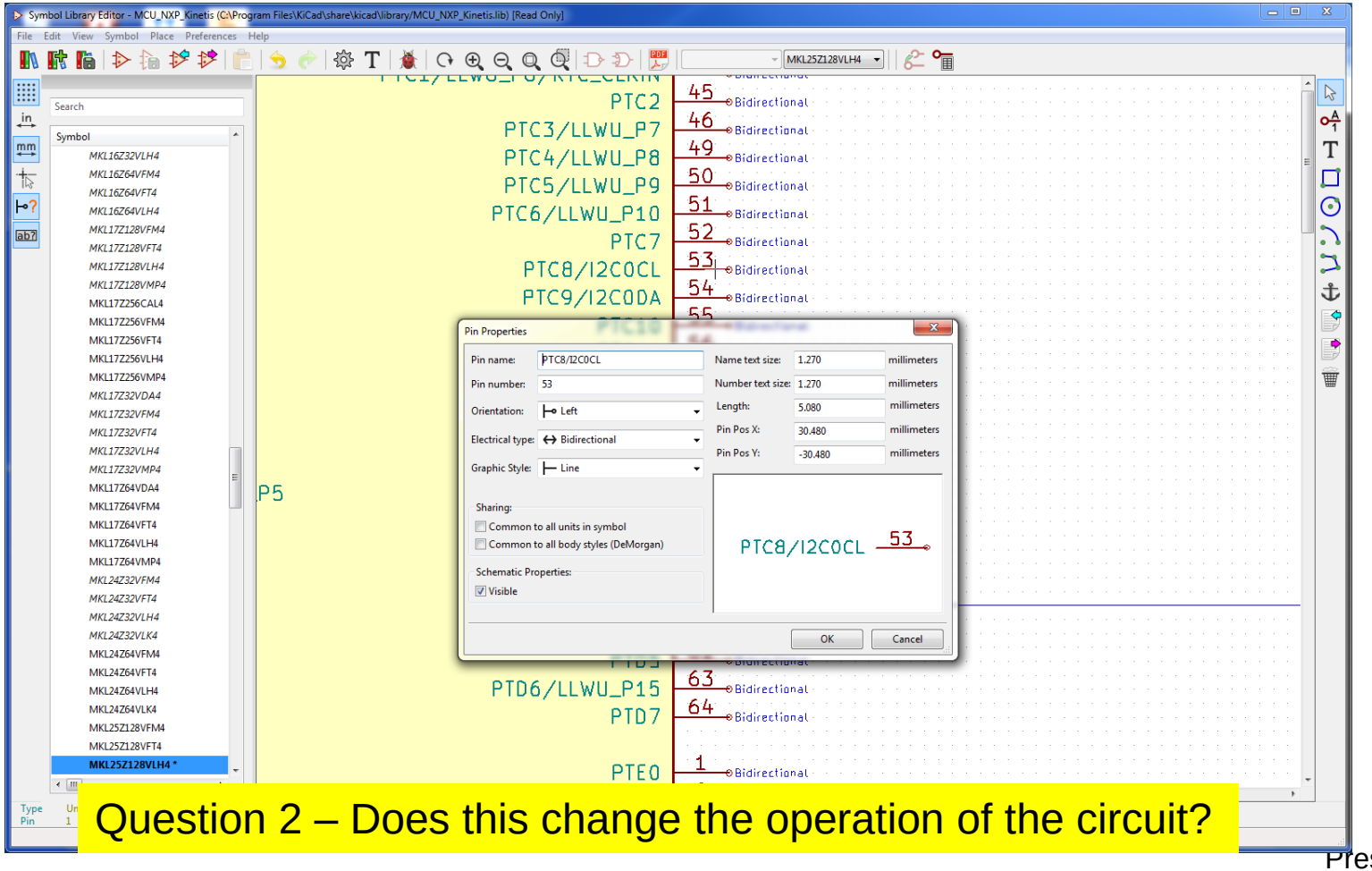
We Need a I²C Port

				LLWU_P10			LLWU_P10	
64	52	40	28	PTC7	CMP0_IN1	CMP0_IN1	PTC7	SPI0_MISO
65	53	—	—	PTC8	CMP0_IN2	CMP0_IN2	PTC8	I2C0_SCL
66	54	—	—	PTC9	CMP0_IN3	CMP0_IN3	PTC9	I2C0_SDA
67	55	—	—	PTC10	DISABLED		PTC10	I2C1_SCL
68	56	—	—	PTC11	DISABLED		PTC11	I2C1_SDA
69	—	—	—	PTC12	DISABLED		PTC12	

PTC8 and PTC9 are Mis-labeled

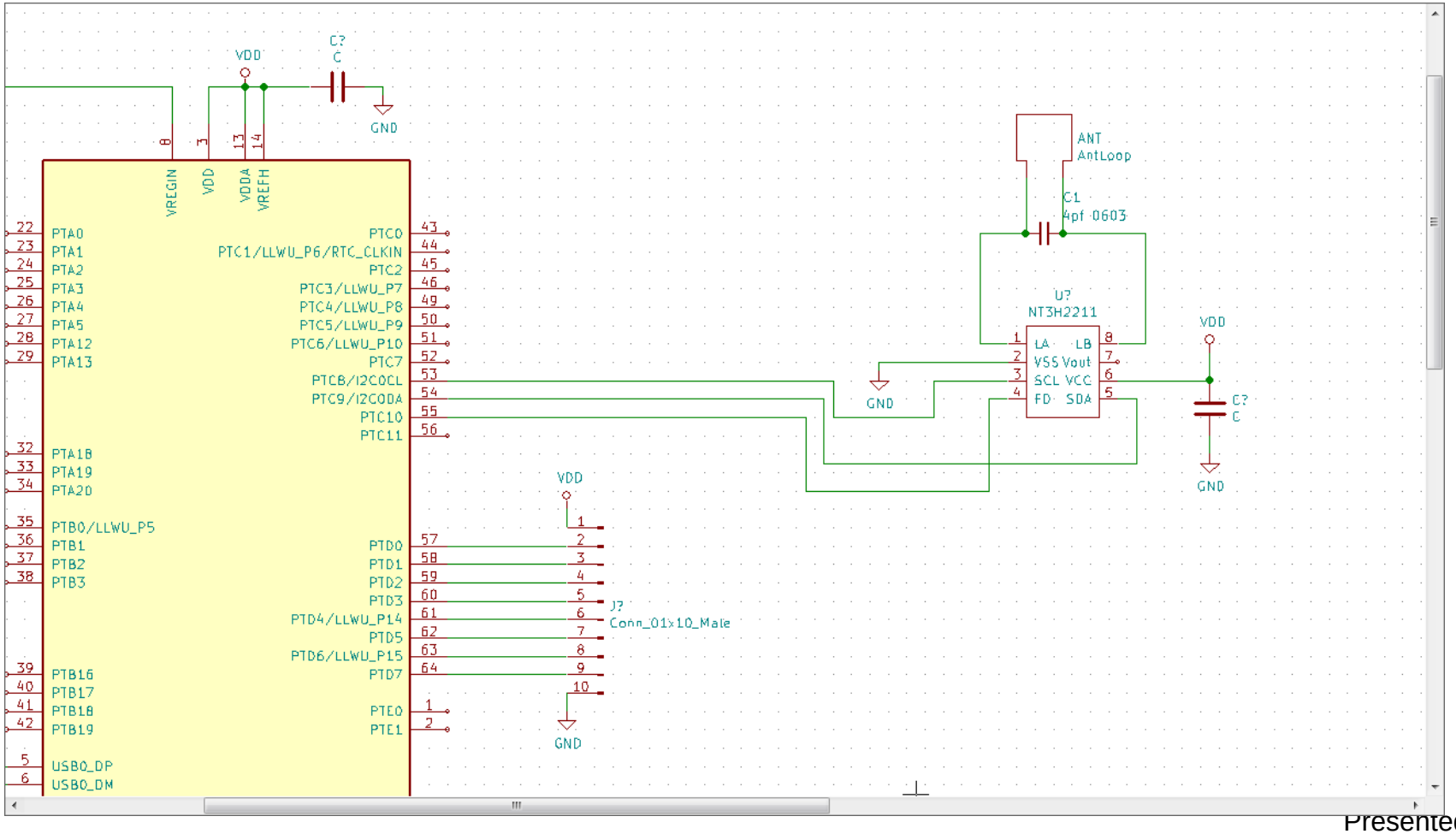


We add the functions to pin description



Question 2 – Does this change the operation of the circuit?

Saved changed KL25 to our library

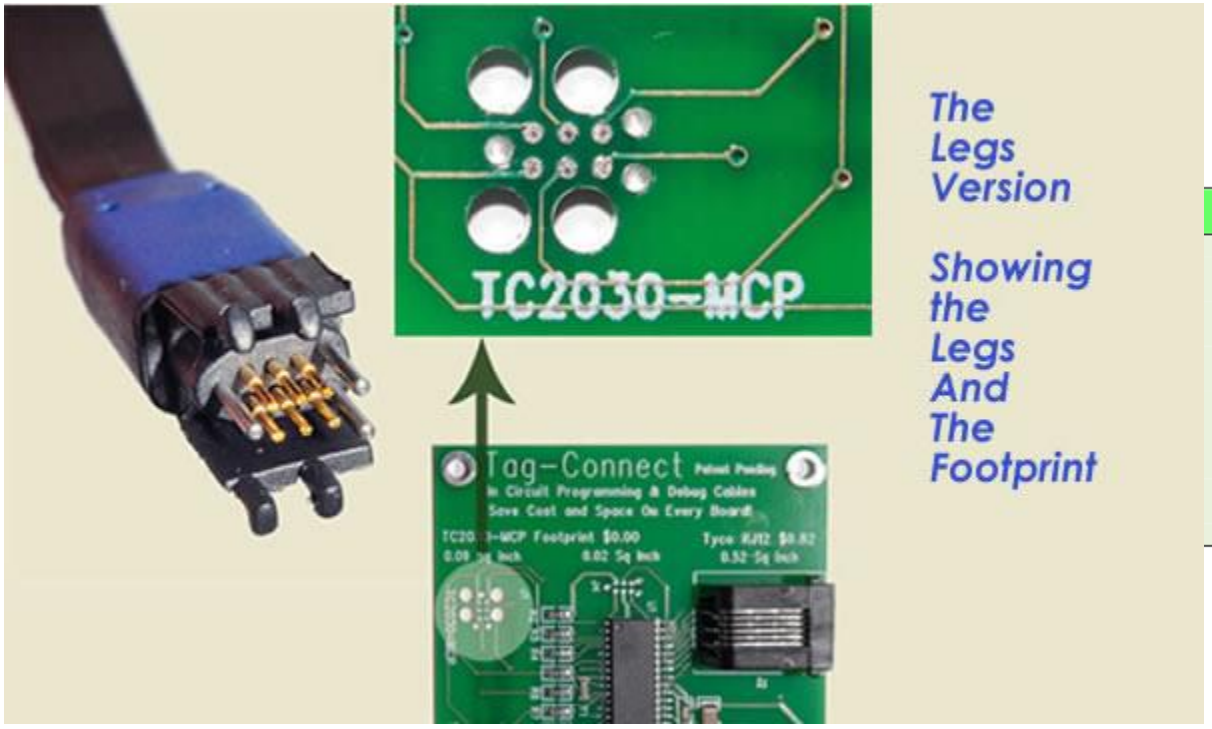


We Need to Program our KL25

- Cortex processors use the Serial Wire Debug (SWD)
- As this only needs 2 pins (plus ground and Vdd reference) and optional reset, we can use a 6-pin standard interface
- I use a connectorless system called Tag-Connect

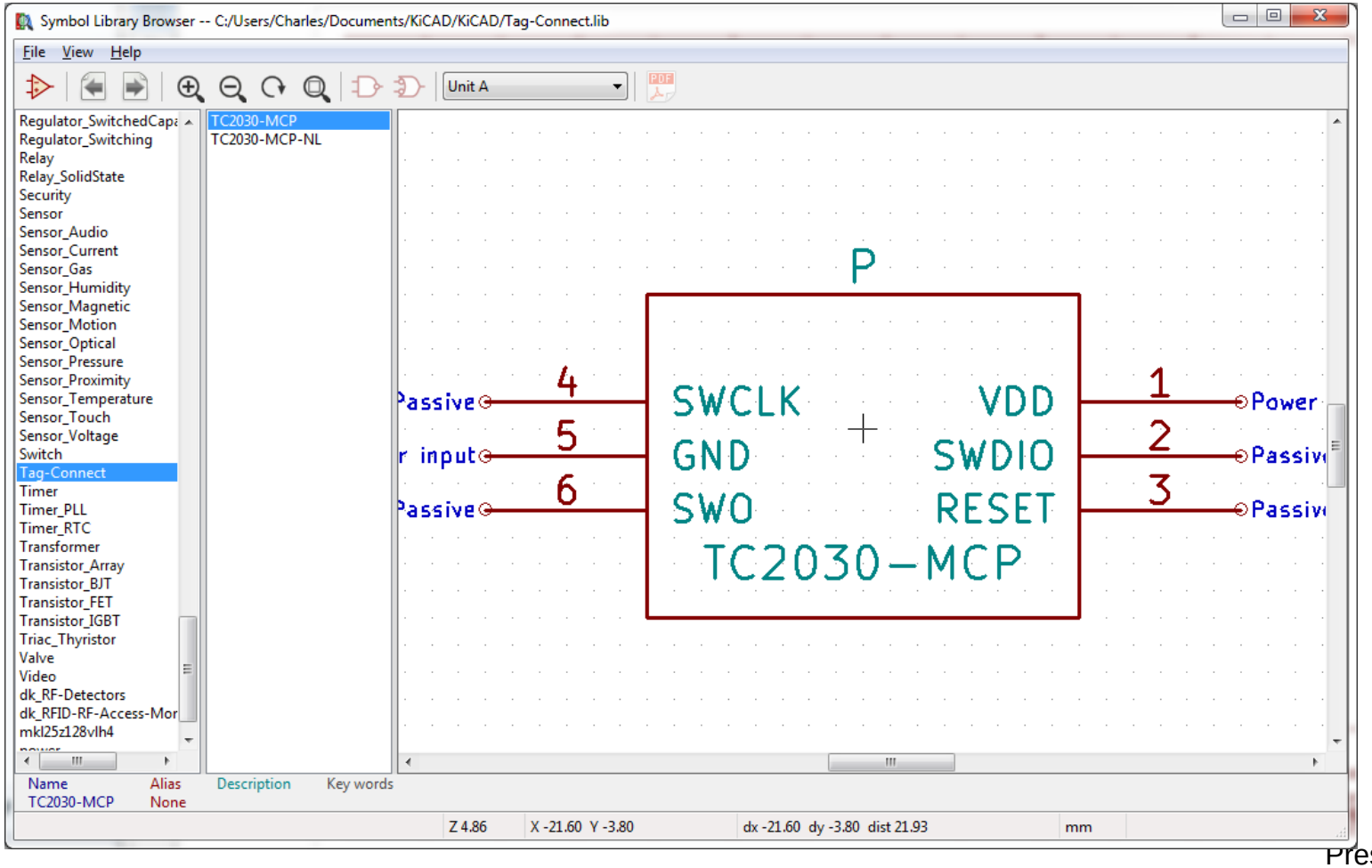
Tag-connect.com

The Important Info

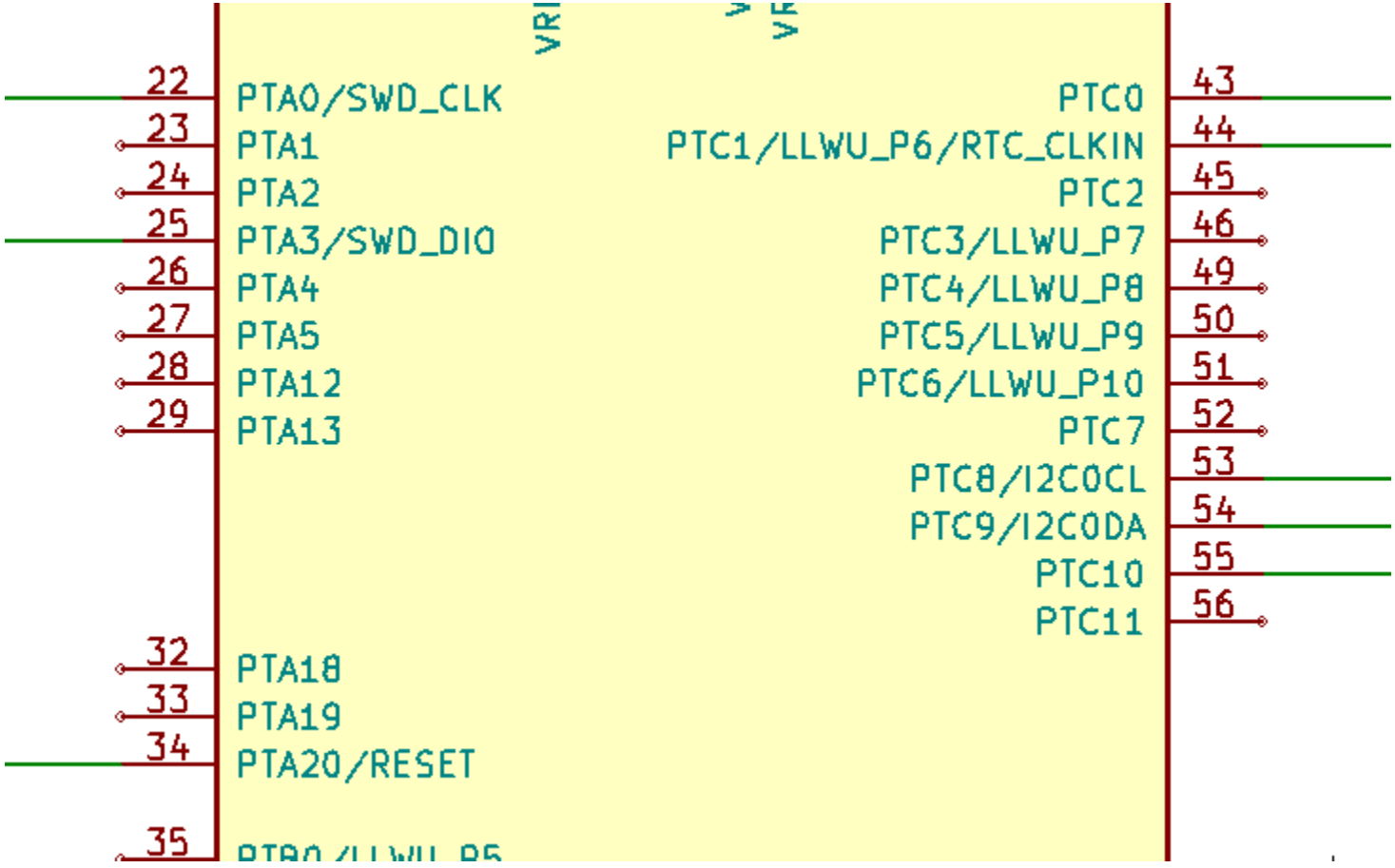


6-Pin TC2030 Footprint
1 VCC
2 SWDIO / TMS
3 nRESET
4 SWCLK / TCK
5 GND (also connected to GNDDetect)
6 SWO / TDO

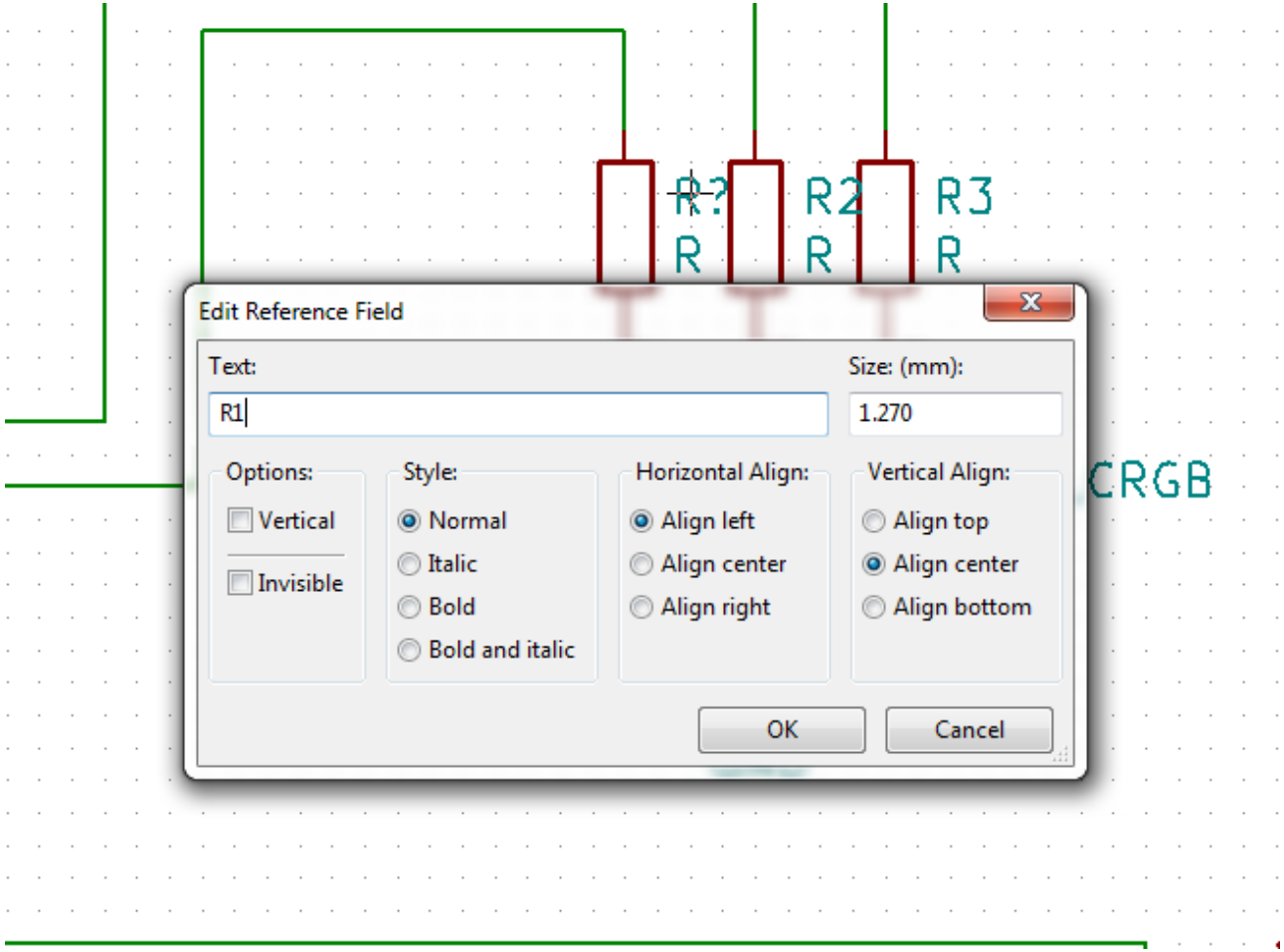
Symbol


Symbol Library Browser -- C:/Users/Charles/Documents/KiCAD/KiCAD/Tag-Connect.lib
File View Help
Unit A
Regulator_SwitchedCapa
Regulator_Switching
Relay
Relay_SolidState
Security
Sensor
Sensor_Audio
Sensor_Current
Sensor_Gas
Sensor_Humidity
Sensor_Magnetic
Sensor_Motion
Sensor_Optical
Sensor_Pressure
Sensor_Proximity
Sensor_Temperature
Sensor_Touch
Sensor_Voltage
Switch
Tag-Connect
Timer
Timer_PLL
Timer_RTC
Transformer
Transistor_Array
Transistor_BJT
Transistor_FET
Transistor_IGBT
Triac_Thyristor
Valve
Video
dk_RF-Detectors
dk_RFID-RF-Access-Mor
mkl25z128vlh4
TC2030-MCP
TC2030-MCP-NL
P
Passive
r input
Passive
4
5
6
SWCLK
GND
SWO
VDD
SWDIO
RESET
TC2030-MCP
1
2
3
Power
Passive
Passive
Name
Alias
Description
Key words
TC2030-MCP
None
Z 4.86
X -21.60 Y -3.80
dx -21.60 dy -3.80 dist 21.93
mm

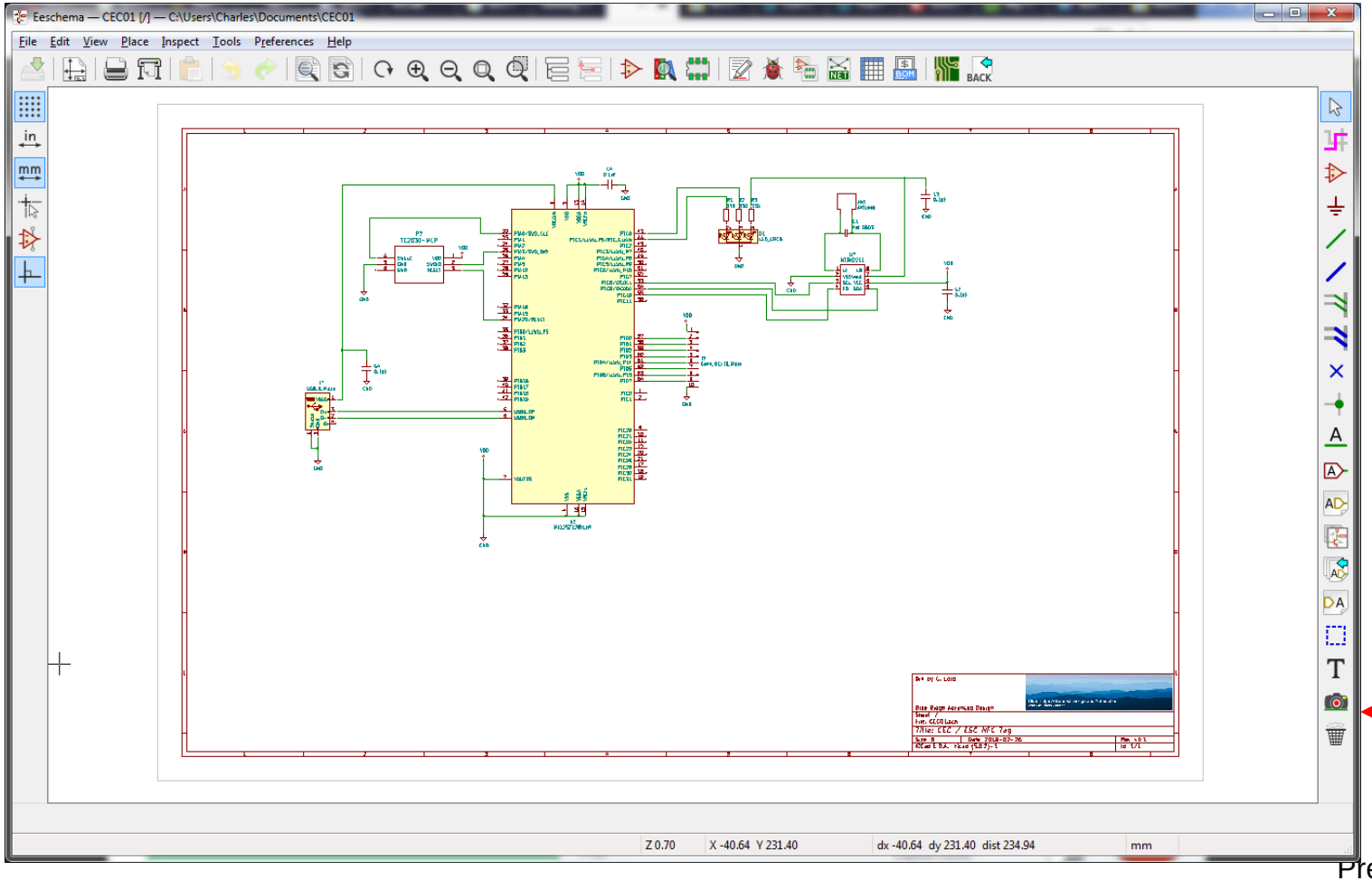
Added SWD Labels



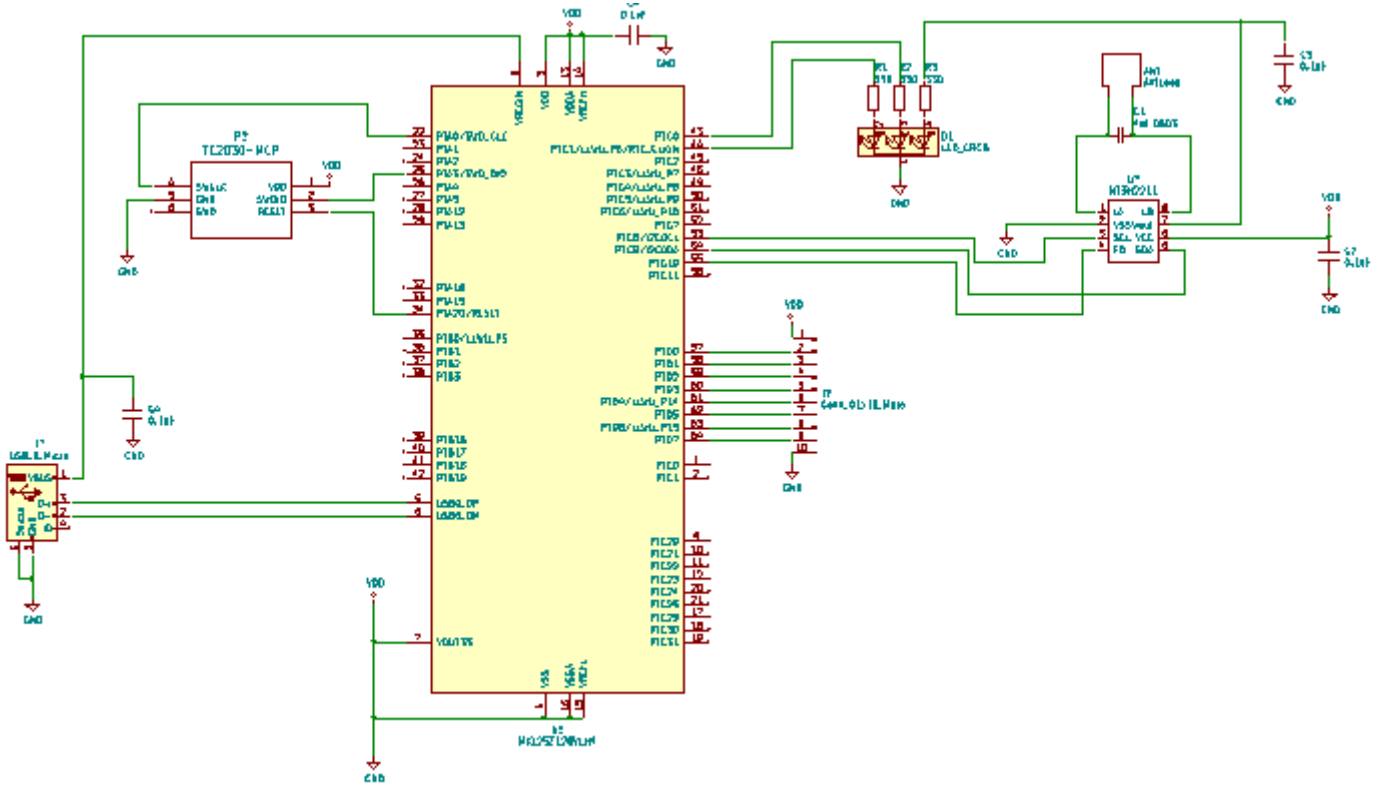
Generic R and C, Edit Parameters



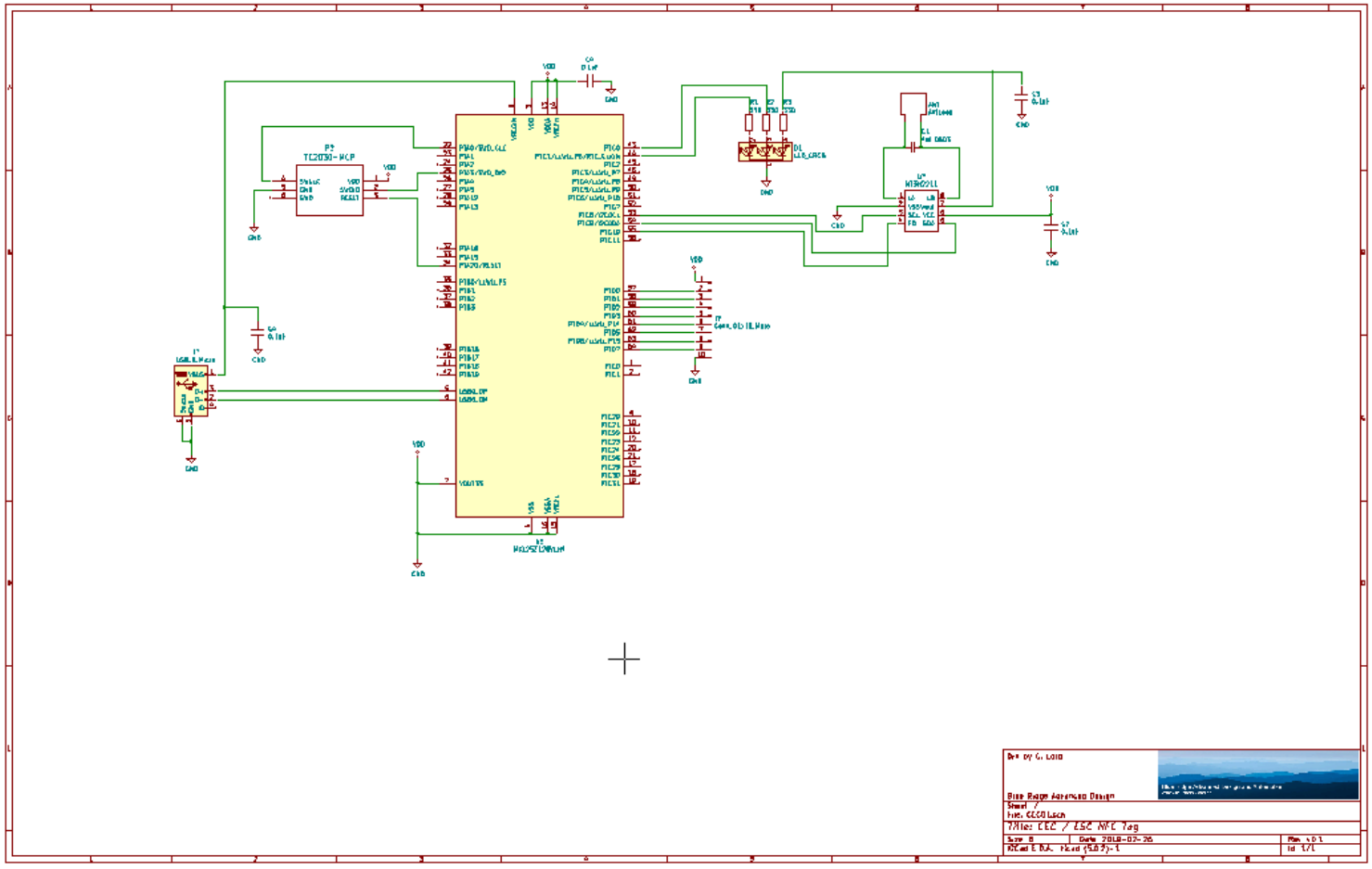
Added a bitmap (logo)



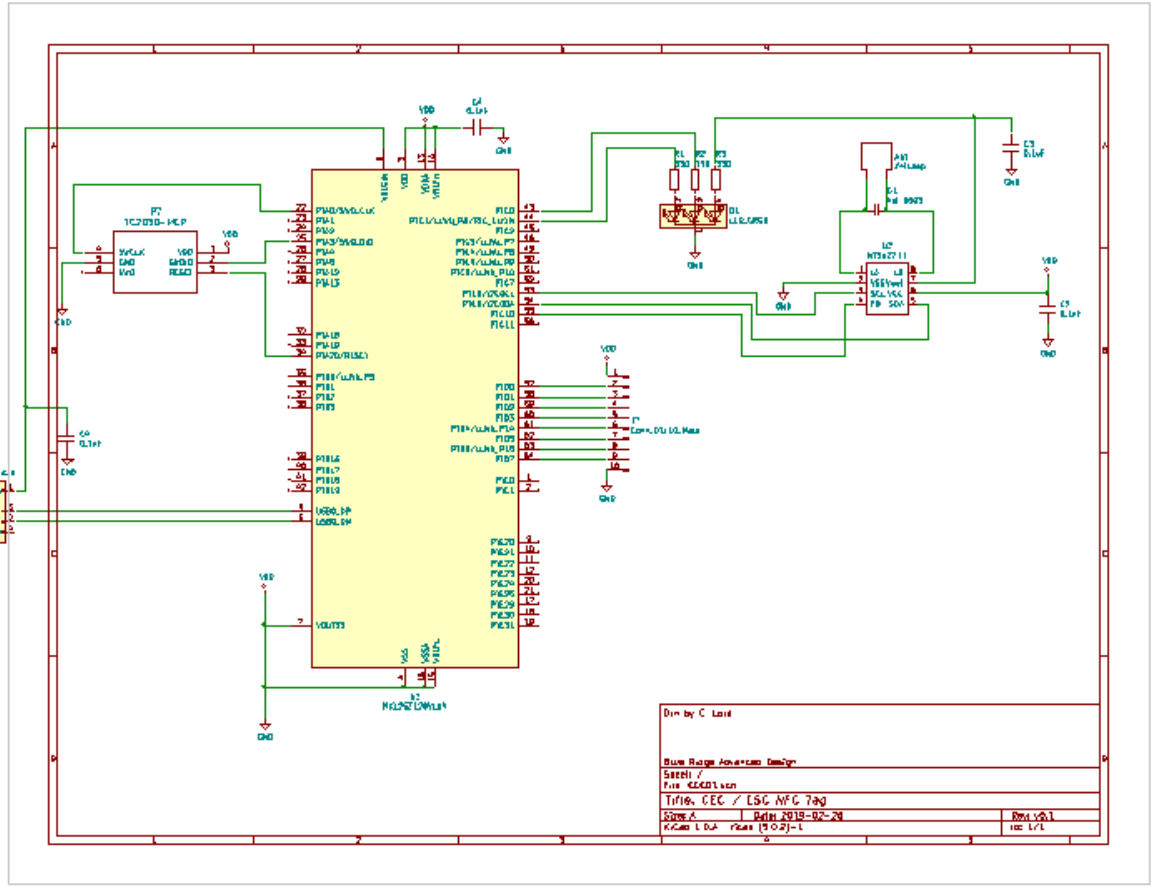
Completed Circuit



A little small on B paper



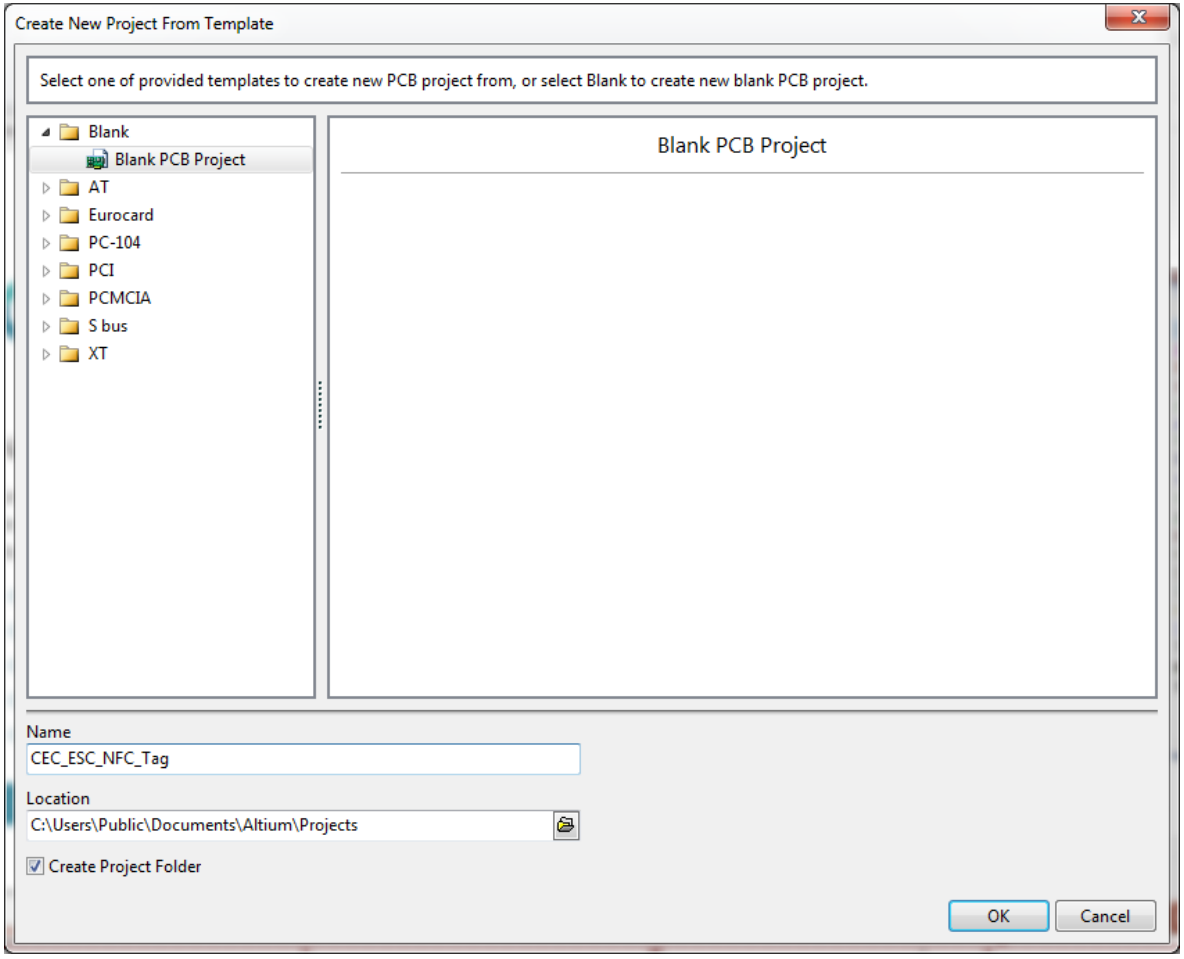
Doesn’t fit 8-1/2 x 11

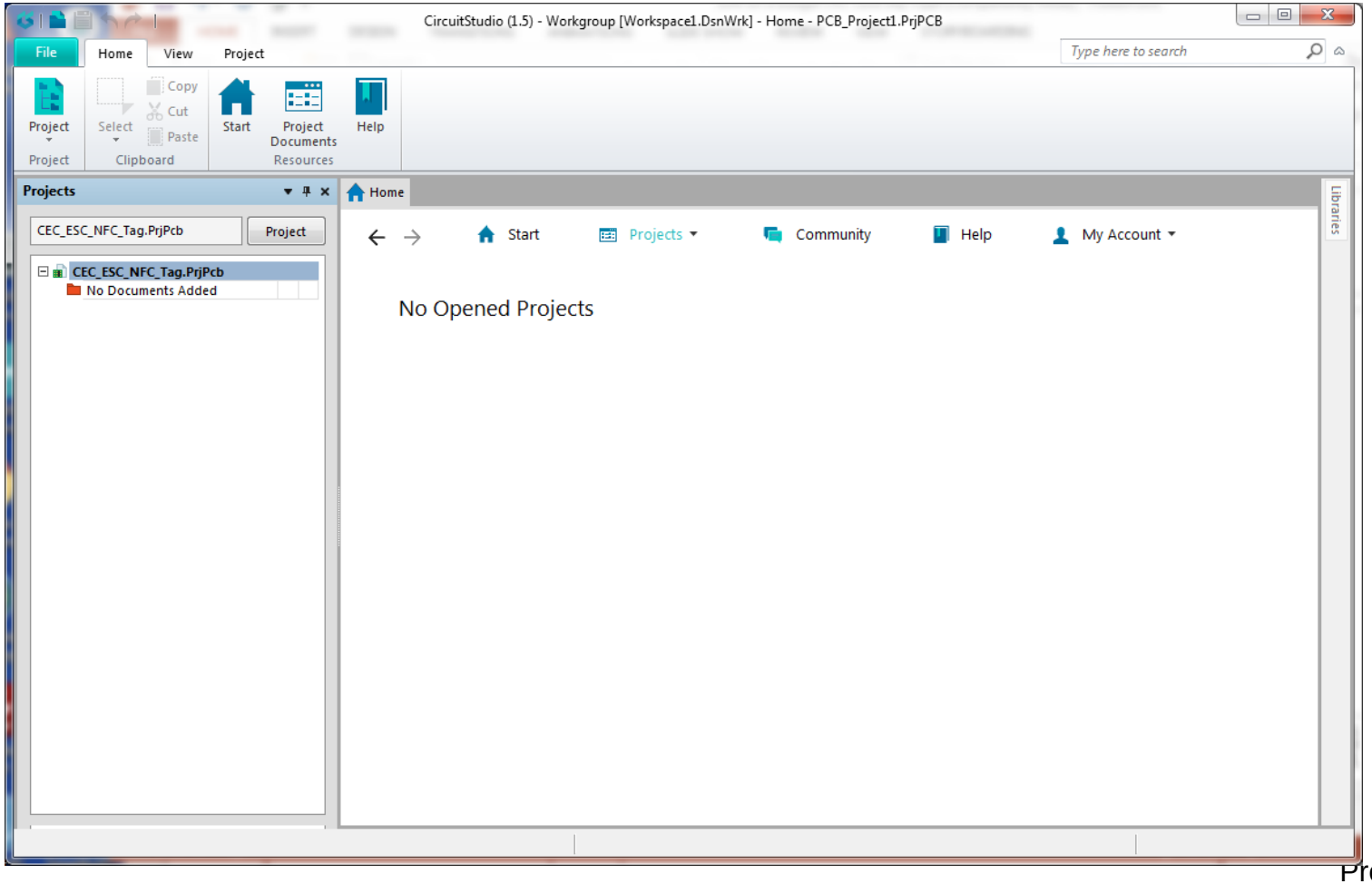


CircuitStudio

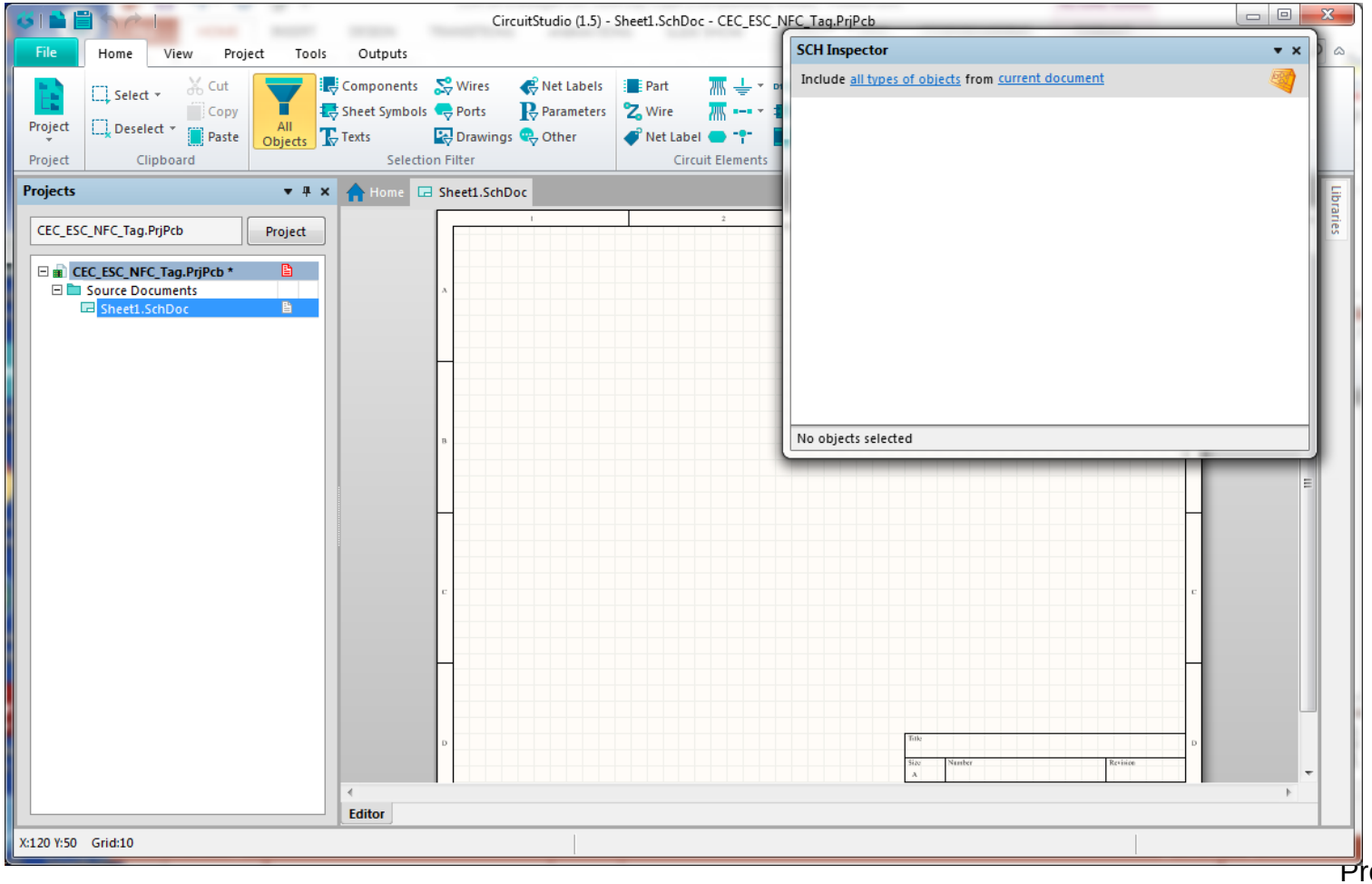
- Let’s catch up with where we are in KiCad

Click on File, New PCB Project

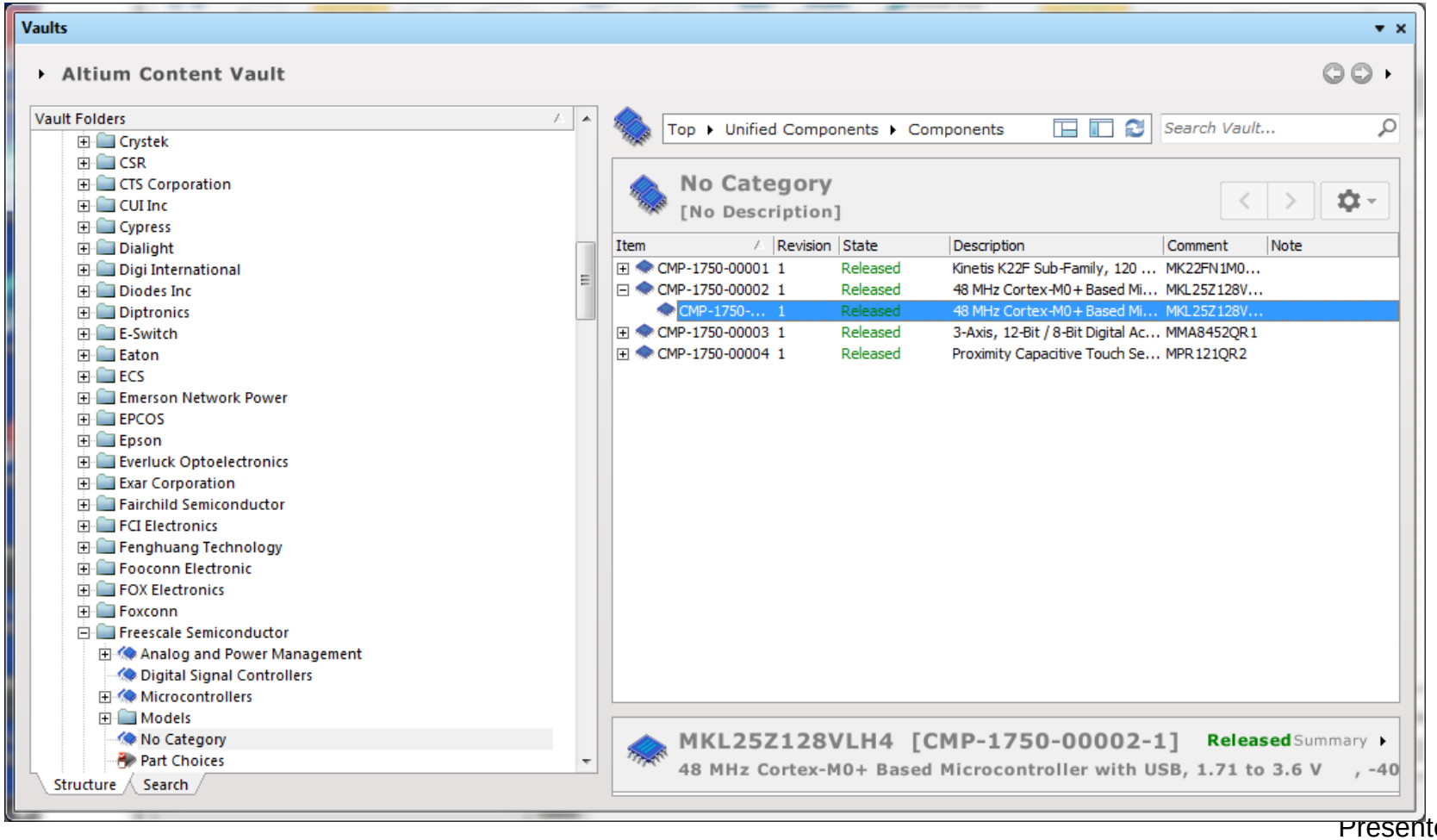



CircuitStudio (1.5) - Workgroup [Workspace1.DsnWrk] - Home - PCB_Project1.PrjPCB
File
Home
View
Project
Type here to search
Project
Select
Copy
Cut
Paste
Start
Project Documents
Help
Clipboard
Resources
Projects
CEC_ESC_NFC_Tag.PrjPcb
Project
CEC_ESC_NFC_Tag.PrjPcb
No Documents Added
Home
Start
Projects
Community
Help
My Account
No Opened Projects
Libraries

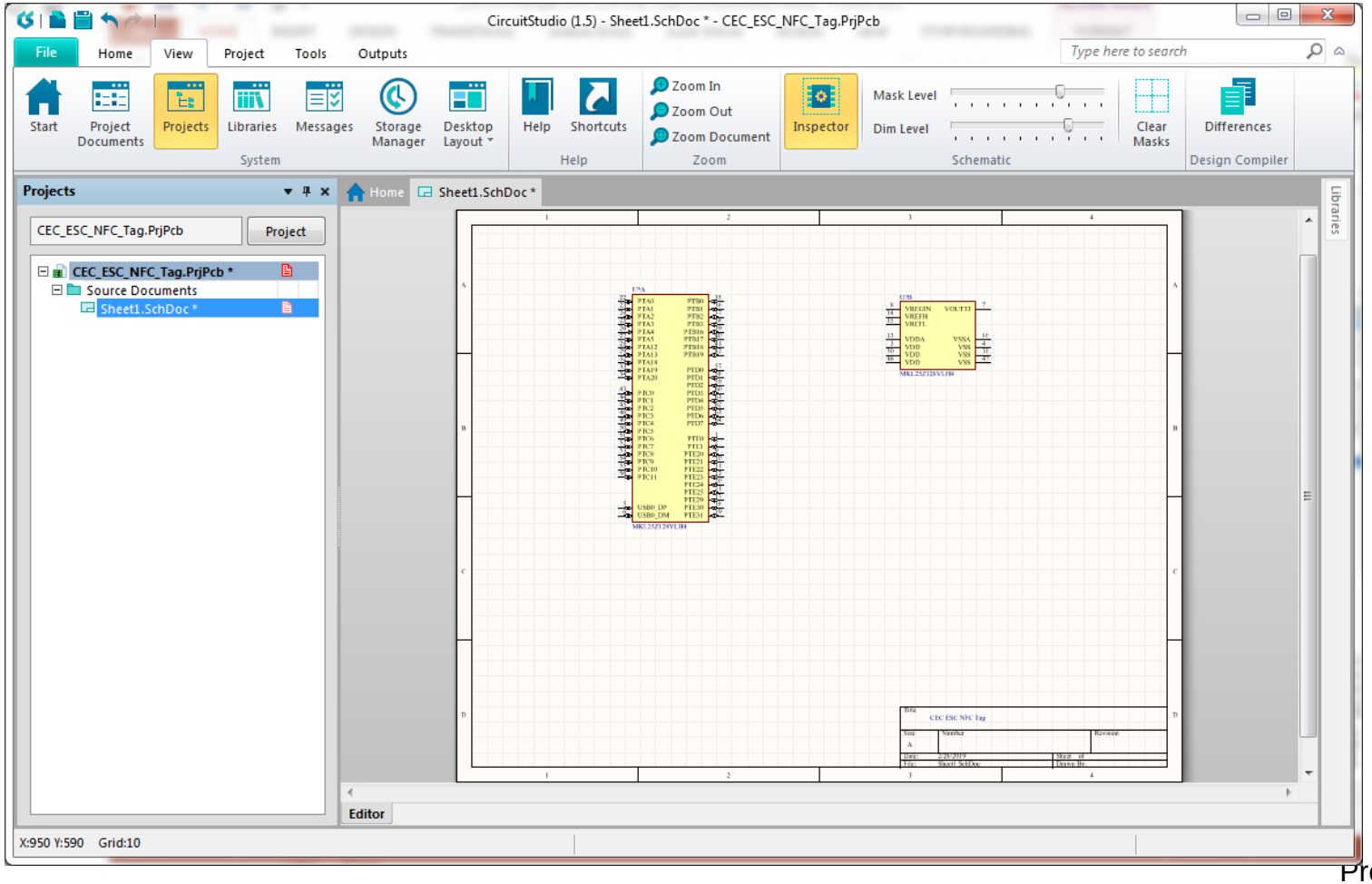
Project, Add New Schematic



Vault Explorer, KL25Z?

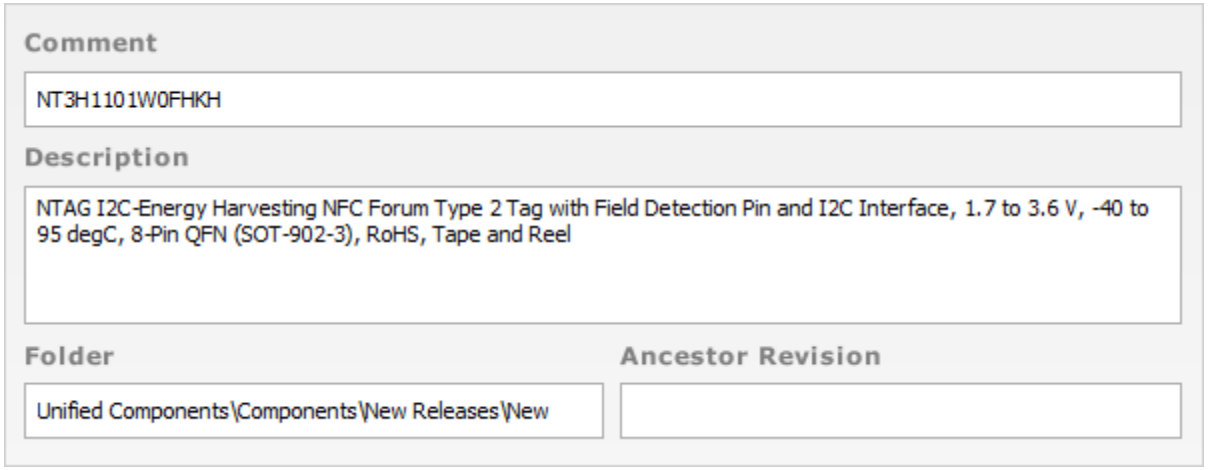


Part A and B

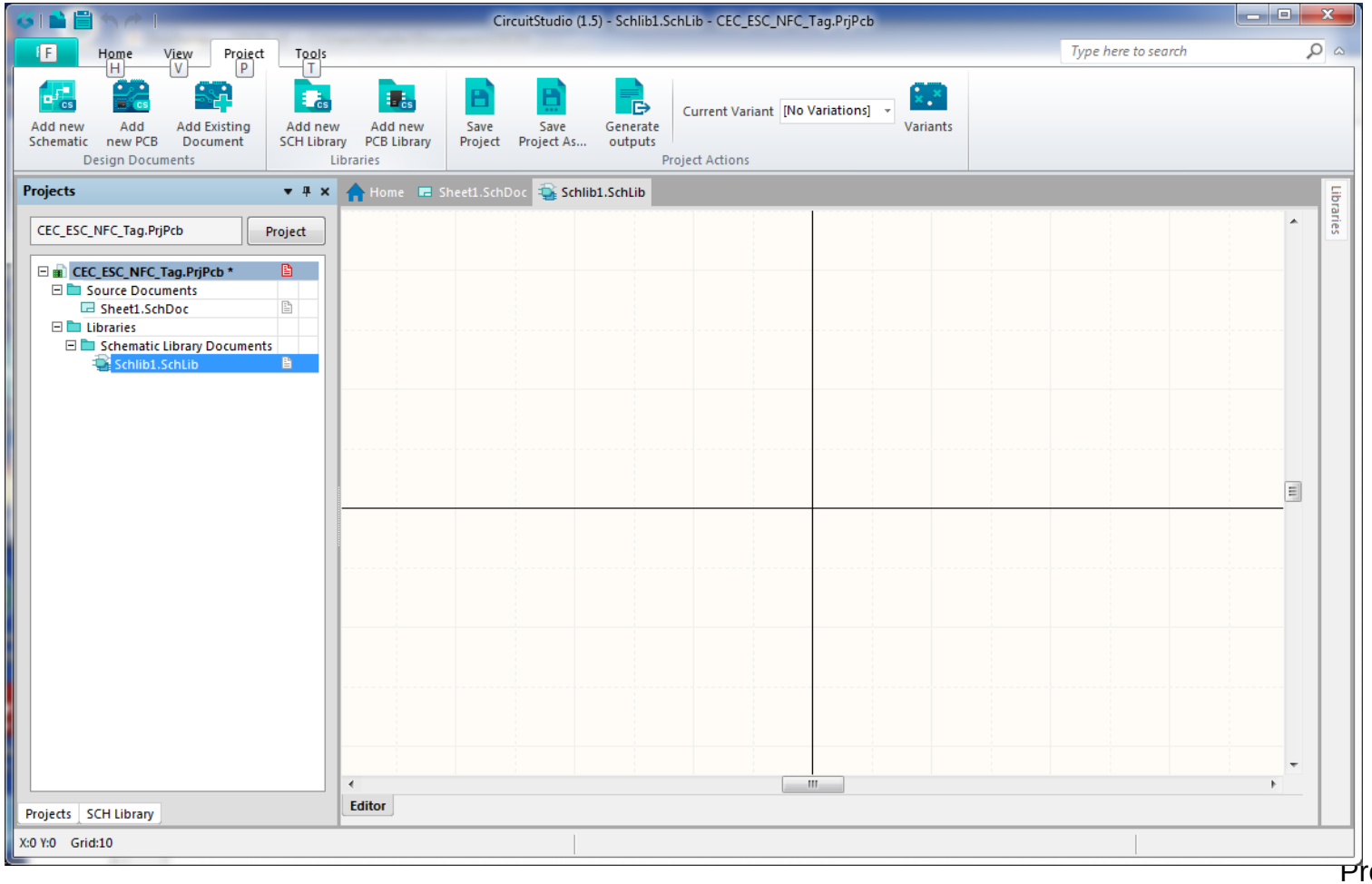

CircuitStudio (1.5) - Sheet1.SchDoc * - CEC_ESC_NFC_Tag.PrjPcb
File
Home
View
Project
Tools
Outputs
Type here to search
Start
Project Documents
Projects
Libraries
Messages
Storage Manager
Desktop Layout
System
Help
Shortcuts
Help
Zoom In
Zoom Out
Zoom Document
Zoom
Inspector
Mask Level
Dim Level
Clear Masks
Schematic
Differences
Design Compiler
Projects
CEC_ESC_NFC_Tag.PrjPcb
Project
CEC_ESC_NFC_Tag.PrjPcb *
Source Documents
Sheet1.SchDoc *
Home
Sheet1.SchDoc *
Libraries
Editor
X:950 Y:590 Grid:10

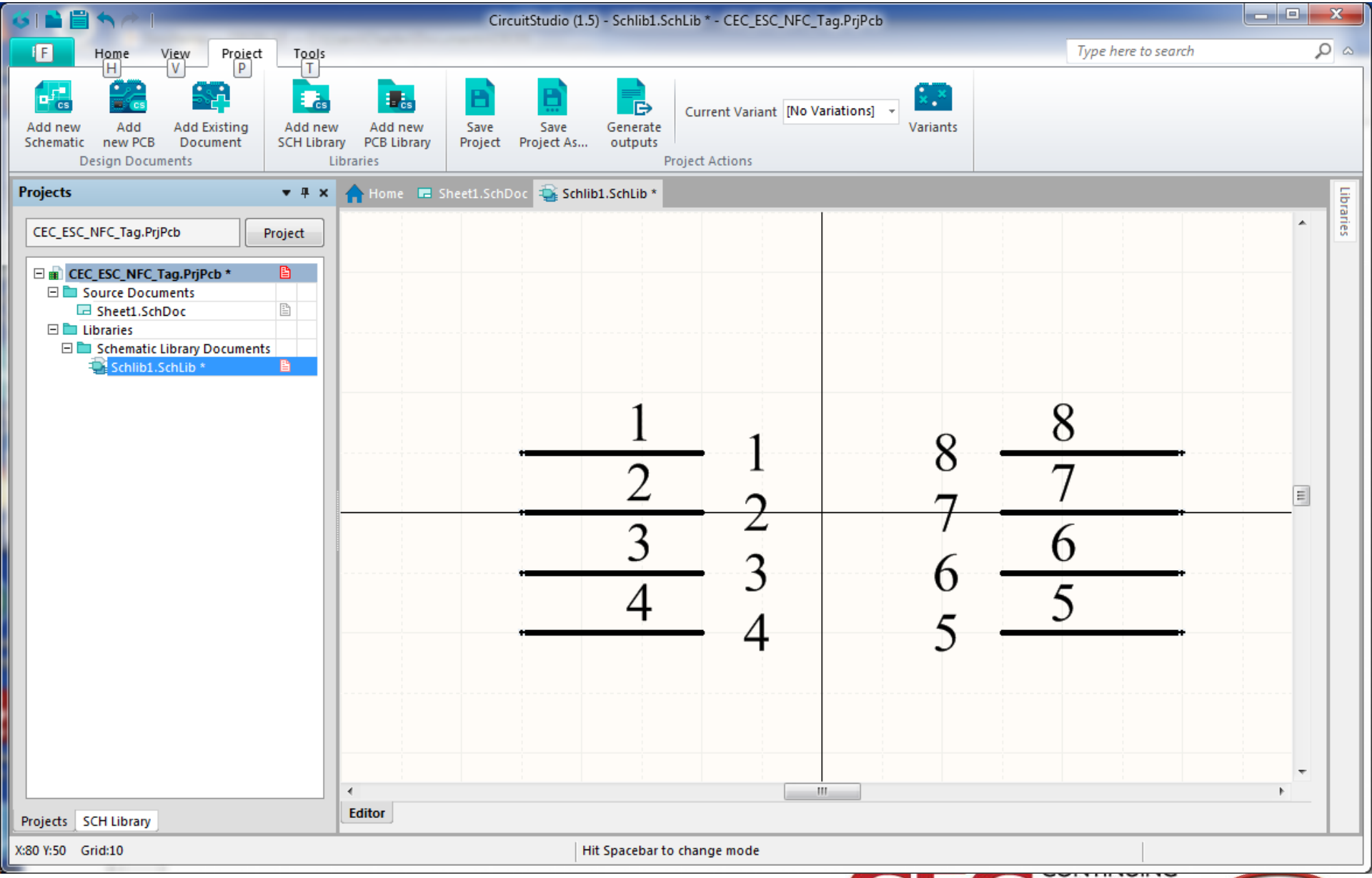
Now the NT3H2211

- Going to the vault for the NTAG part, we see it isn't there. We do find a similar part and look to see if we can use it:

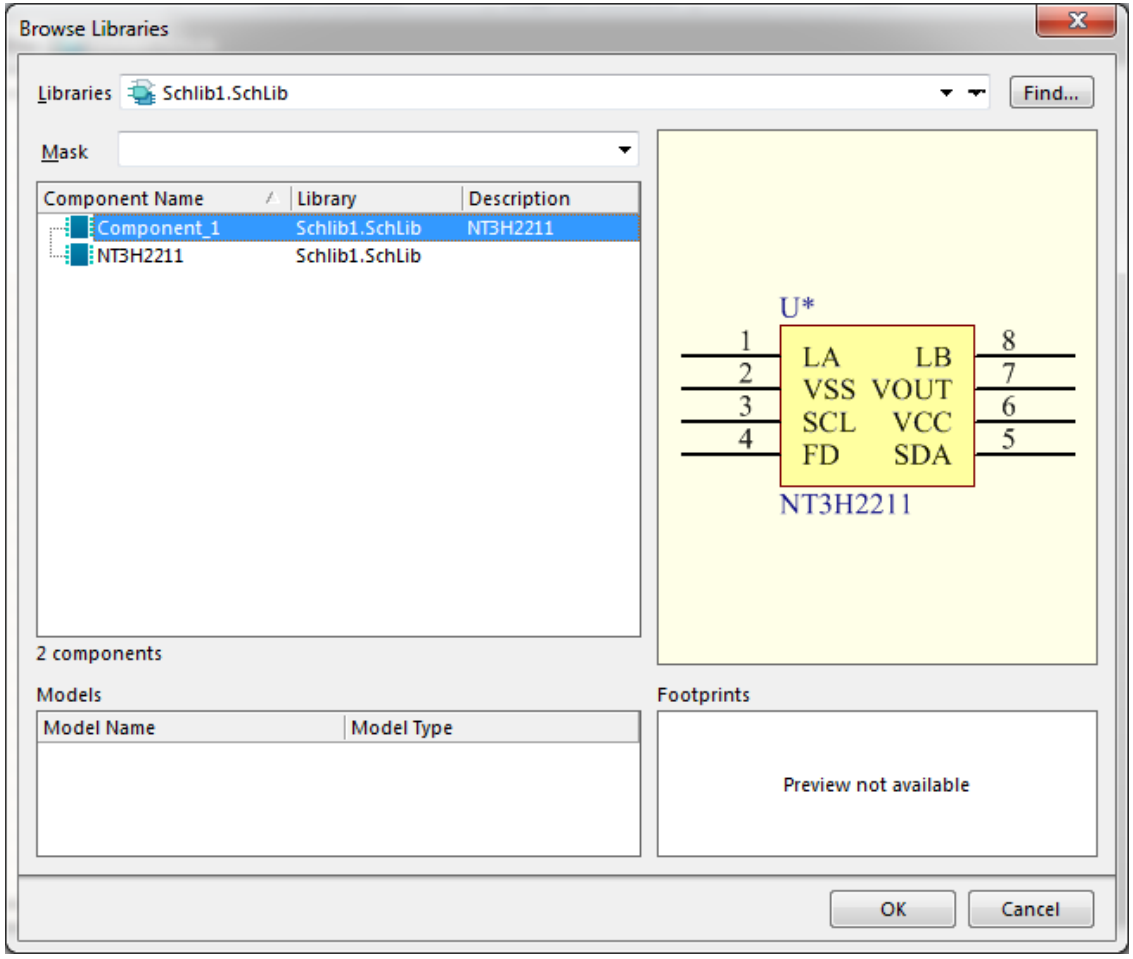


We Need to Create

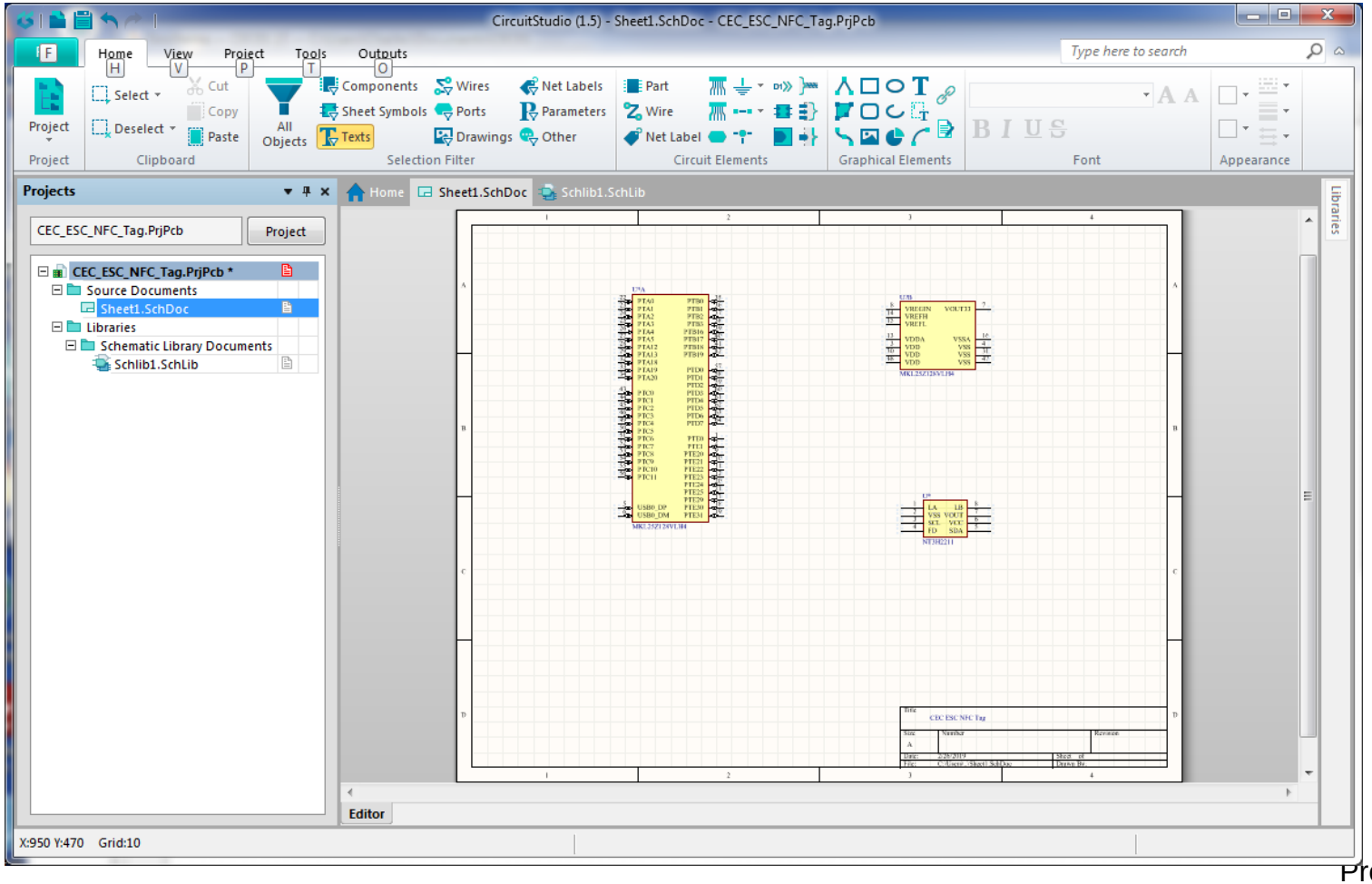



CircuitStudio (1.5) - Schlib1.SchLib * - CEC_ESC_NFC_Tag.PrjPcb
Home
View
Project
Tools
Type here to search
Add new Schematic
Add new PCB
Add Existing Document
Design Documents
Add new SCH Library
Add new PCB Library
Libraries
Save Project
Save Project As...
Generate outputs
Project Actions
Current Variant
[No Variations]
Variants
Projects
CEC_ESC_NFC_Tag.PrjPcb
Project
CEC_ESC_NFC_Tag.PrjPcb *
Source Documents
Sheet1.SchDoc
Libraries
Schematic Library Documents
Schlib1.SchLib *
Home
Sheet1.SchDoc
Schlib1.SchLib *
Libraries
1 2 3 4
1 2 3 4
8 7 6 5
8 7 6 5
Editor
Projects
SCH Library
X:80 Y:50 Grid:10
Hit Spacebar to change mode

Finished Part



IC’s In Place


CircuitStudio (1.5) - Sheet1.SchDoc - CEC_ESC_NFC_Tag.PrjPcb
Home
View
Project
Tools
Outputs
Type here to search
Select
Deselect
Cut
Copy
Paste
All Objects
Components
Sheet Symbols
Texts
Wires
Ports
Drawings
Net Labels
Parameters
Other
Part
Wire
Net Label
Projects
CEC_ESC_NFC_Tag.PrjPcb
Project
CEC_ESC_NFC_Tag.PrjPcb *
Source Documents
Sheet1.SchDoc
Libraries
Schematic Library Documents
Schlib1.SchLib
Home
Sheet1.SchDoc
Schlib1.SchLib
Libraries
Editor
X:950 Y:470
Grid:10

Tomorrow!

- A peek at the finished CircuitStudio schematic
- We assign footprints to parts
- We check for proper netlist
- Let's design a PC Board!

Question 3 – What KiCad hot key rotates a part?

This Week's Agenda

2/25 Introduction to EDA and Our Tools

2/26 Starting at the Beginning

2/27 Creating Our Schematic

2/28 Creating Our PCB Layout

3/1 Our Finished Board

Please stick around as I answer your questions!

- Please give me a moment to scroll back through the chat window to find your questions
- I will stay on chat as long as it takes to answer!
- I am available to answer simple questions or to consult (or offer in-house training for your company)
  c.j.lord@ieee.org
  http://www.blueridgetechnc.com
  http://www.linkedin.com/in/charleslord
  Twitter: @charleslord
  https://www.github.com/bradatraining