

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

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**

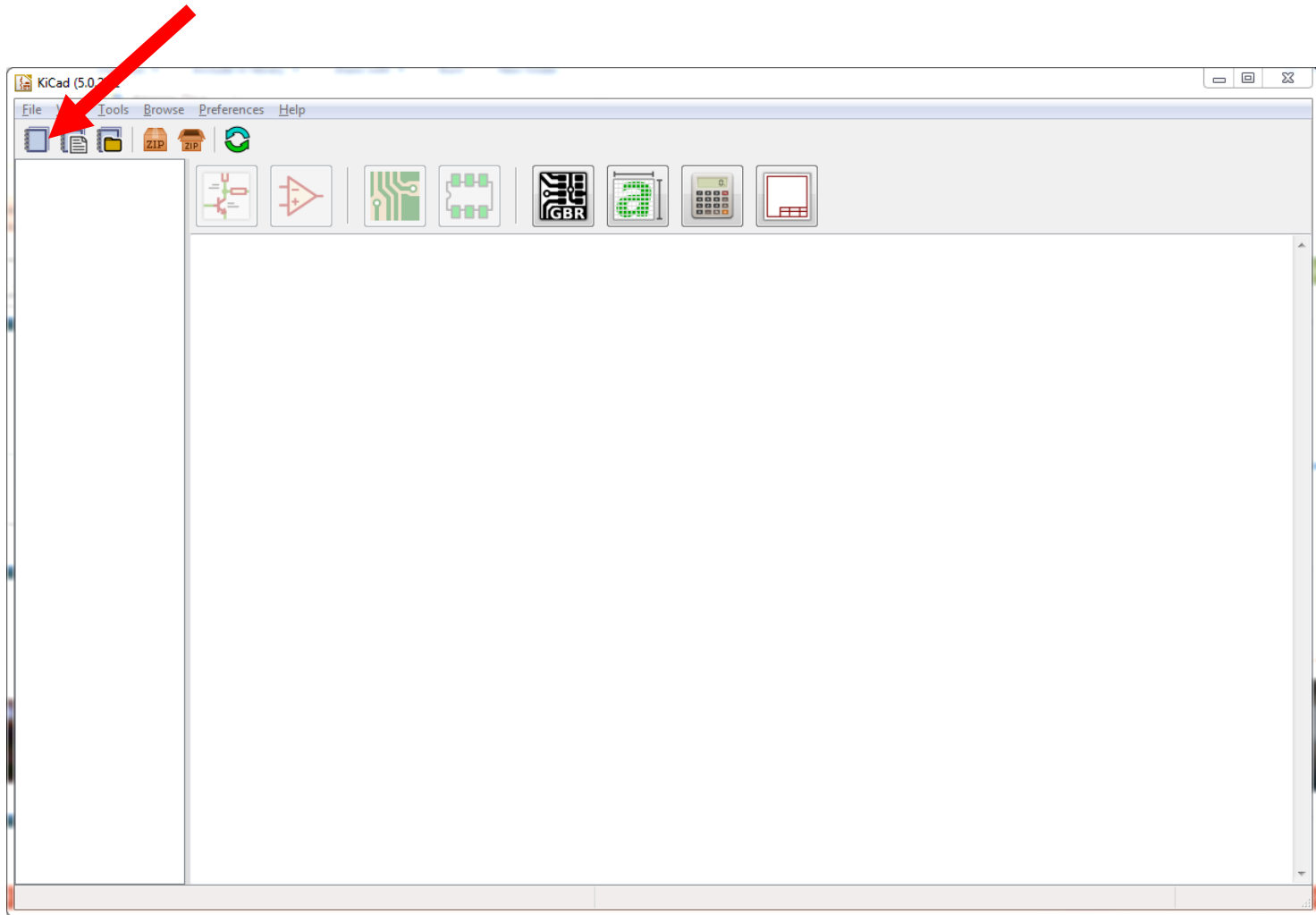
2/28 Creating Our PCB Layout

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



Presented by:

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

Digi-Key EDA & Design Tools – *Think It...Analyze it...Design It™*

No matter where you are in the design cycle, Digi-Key has the right tool to help you meet your design challenges head-on. From the industry's most flexible online diagramming tool that enables true back-of-the-napkin to bill of materials (BOM to BOM) functionality, through analog/power simulation, all the way to the most robust professional full PCB schematic capture and layout functionality, Digi-Key's unparalleled design tool solutions are ready to help bring your designs to fruition quicker and easier than ever before. Each tool is uniquely paired with access to Digi-Key's inventory of the world's largest selection of electronic components and we are committed to offering you the strongest EDA tool portfolio to help you take your designs to the next level.



Scheme-it – Schematic Drawing and Block Diagramming Made Easy.

Scheme-it is an online schematic and block diagramming tool for designing and sharing electronic circuit diagrams. Design at the schematic, block, or board level.

[Learn More »](#)



Introducing... DK IoT Studio

DK IoT Studio provides the fastest and simplest way to create, deploy and manage a complete embedded-to-cloud IoT solution.

[Learn More »](#)



Ultra Librarian – EDA/CAD Model Libraries Made Easy.

Automated and manual footprint, symbol, and 3D model creation with access to database of pre-built library parts that export to most CAD formats.

[Learn More »](#)



SnapEDA – Build circuit boards faster with instant parts.

Download free symbols & footprints for millions of electronic components. The largest source of verified parts.

[Learn More »](#)



Introducing Digi-Key's KiCad Symbol and Footprint Library.

KiCad is an open source Electronic Design Automation (EDA) tool option that offers the functionality needed for most any projects.

[Learn More »](#)



Quadcept – Revolutionary CAD System

Quadcept offers free and paid CAD applications for schematic capture and PCB layout with powerful routing feature.

[Learn More »](#)



A Siemens Business

PADS Maker | PADS MakerPro

Design printed circuit boards from start to finish quickly and easily with PADS® Maker and PADS® MakerPro.

[Learn More »](#)

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.



Get it Now

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



Try it Now

A copy of this library is also available here to [Download Now](#).

Let's get the Digi-Key Library!

digikey / digikey-kicad-library

Watch 80 Star 564 Fork 88

Code Issues 3 Pull requests 4 Projects 0 Wiki Insights

An atomic parts library for Ki-Cad.

171 commits 2 branches 1 release 8 contributors View license

Branch: master New pull request Create new file Upload files Find file Clone or download

Ben Hest fixed usb connector, and updated library data/status

digikey-footprints.pretty	fixed usb connector, and updated library data/status
digikey-symbols	fixed usb connector, and updated library data/status
src	footprint correction
.gitignore	updated gitignore
LICENSE.md	added license
README.md	fixed usb connector, and updated library data/status

Clone with HTTPS Use SSH

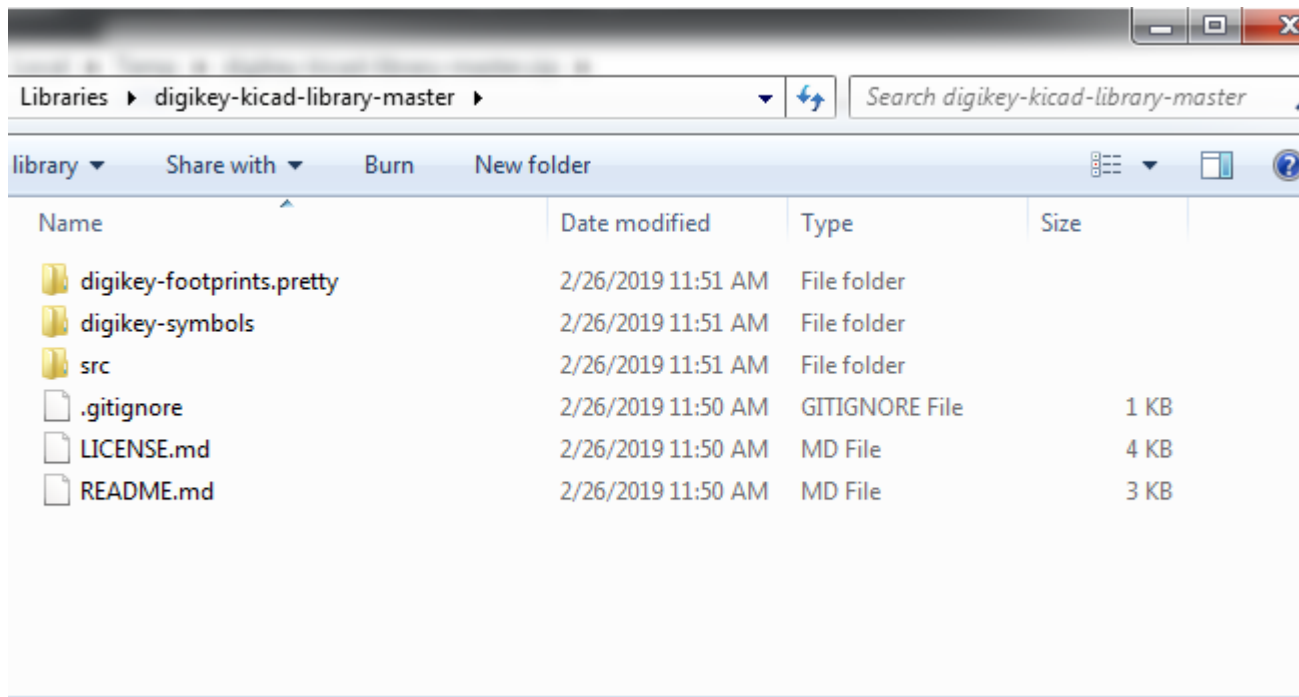
Use Git or checkout with SVN using the web URL.

https://github.com/digikey/digikey-kic

Open in Desktop Download ZIP

Unzip to our Libraries Folder

- Users/public/kicad/libraries/



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

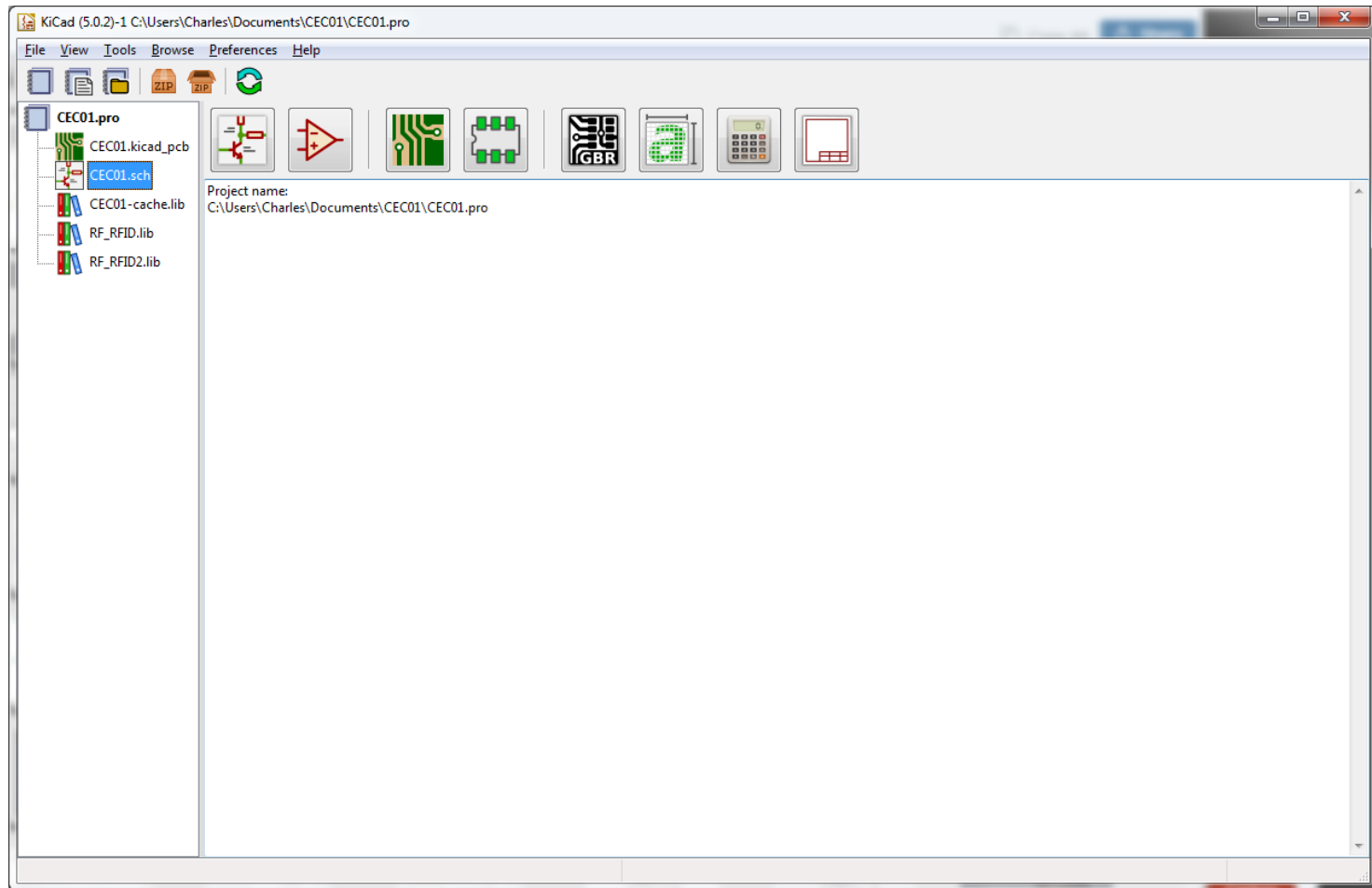
KiCad

HOT

KEYS

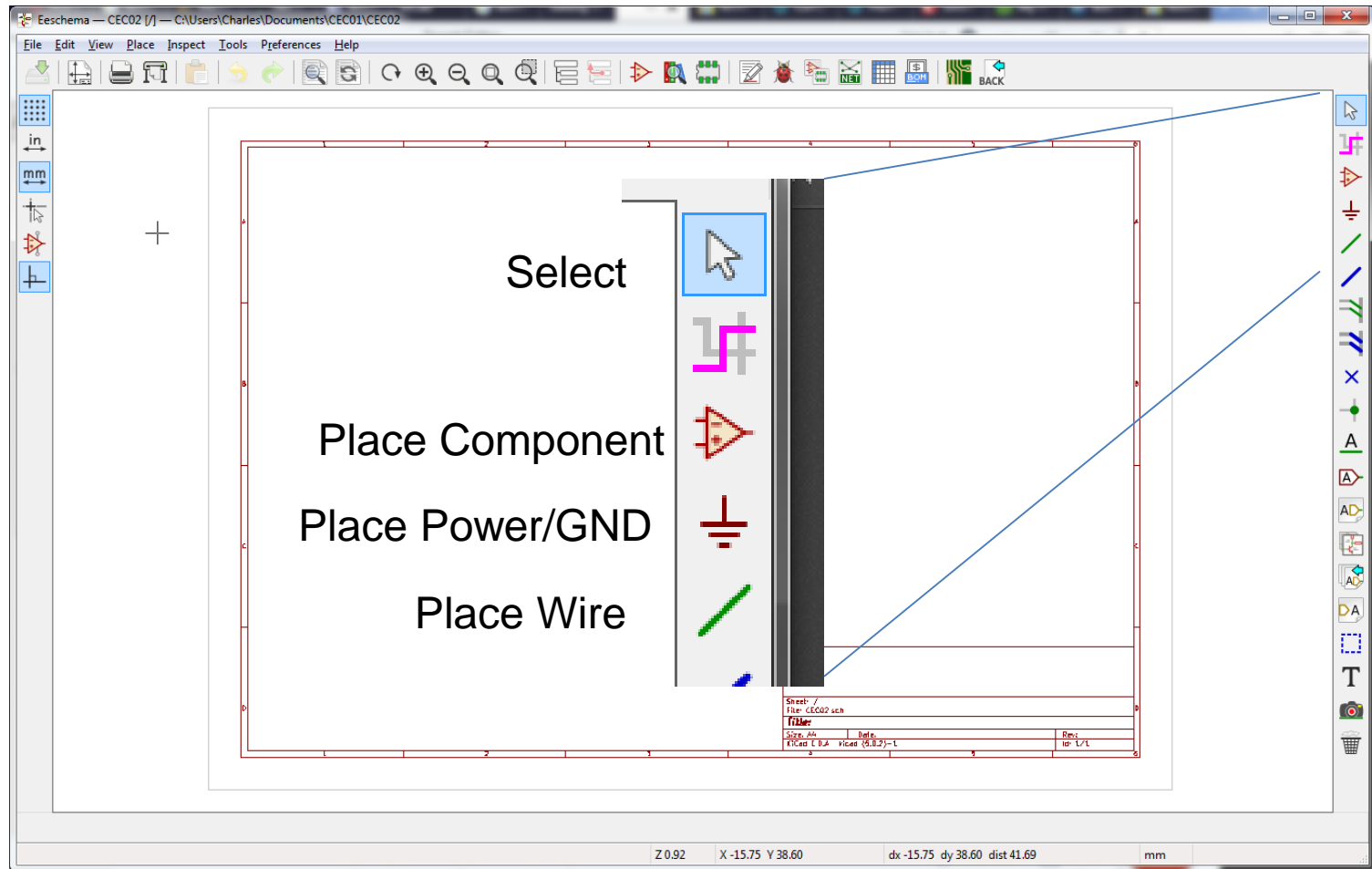
Move Block → Drag Block	Tab
Move Schematic Item	M
Duplicate Symbol or Label	C
Add Symbol	A
Add Power	P
Mirror X	X
Mirror Y	Y
Orient Normal Component	N
Edit Symbol Value	V
Edit Symbol Reference	U
Edit Symbol Footprint	F
Edit with Symbol Editor	Ctrl+E
Begin Wire	W
Begin Bus	B
End Line Wire Bus	K
Add Label	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



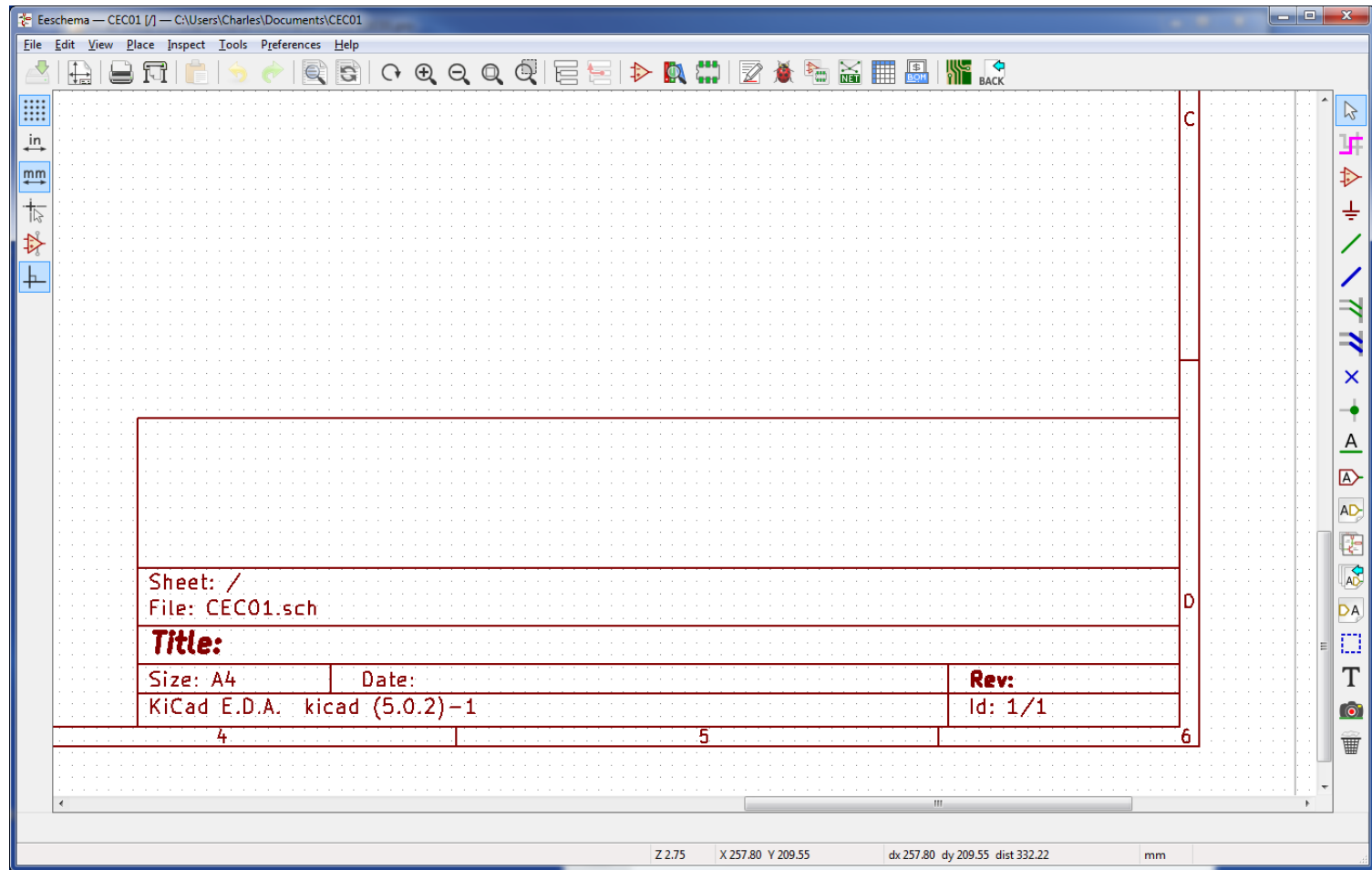
Presented by:

Let's get started!



Presented by:

Title Block



Presented by:

File, Page Settings

Page Settings

Paper

Size: A4 210x297mm

Orientation: Landscape

Custom Size:

Height: 279.40 Width: 431.80

Layout Preview

Title Block Parameters

Number of sheets: 1 Sheet number: 1

Issue Date: 2019-02-26 <<< 2/26/2019 Export to other sheets

Revision: v0.1 Export to other sheets

Title: CEC / ESC NFC Tag Export to other sheets

Company: Blue Ridge Advanced Design Export to other sheets

Comment1: Export to other sheets

Comment2: Export to other sheets

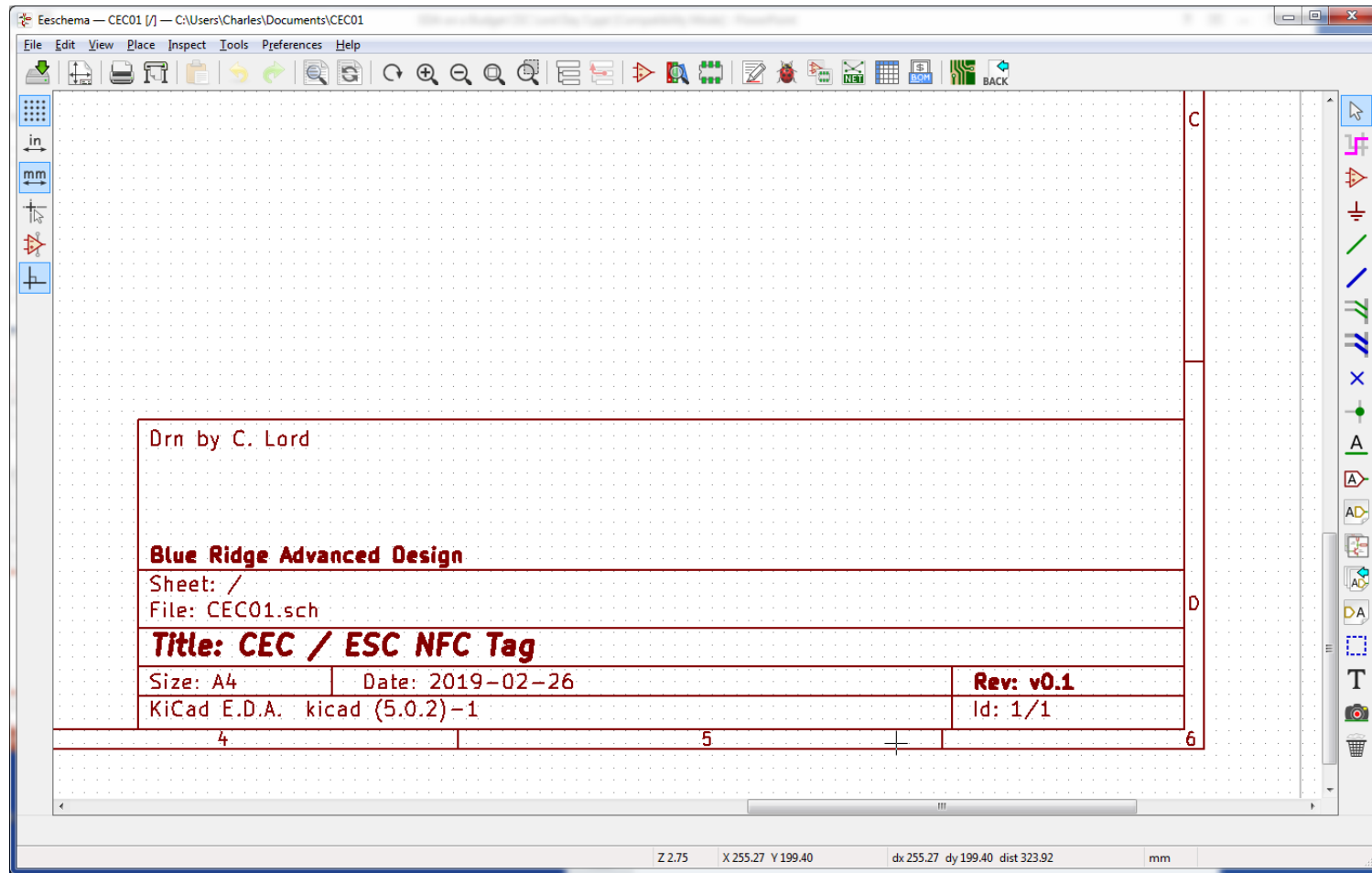
Comment3: Export to other sheets

Comment4: Drn by C. Lord Export to other sheets

Page layout description file: Browse

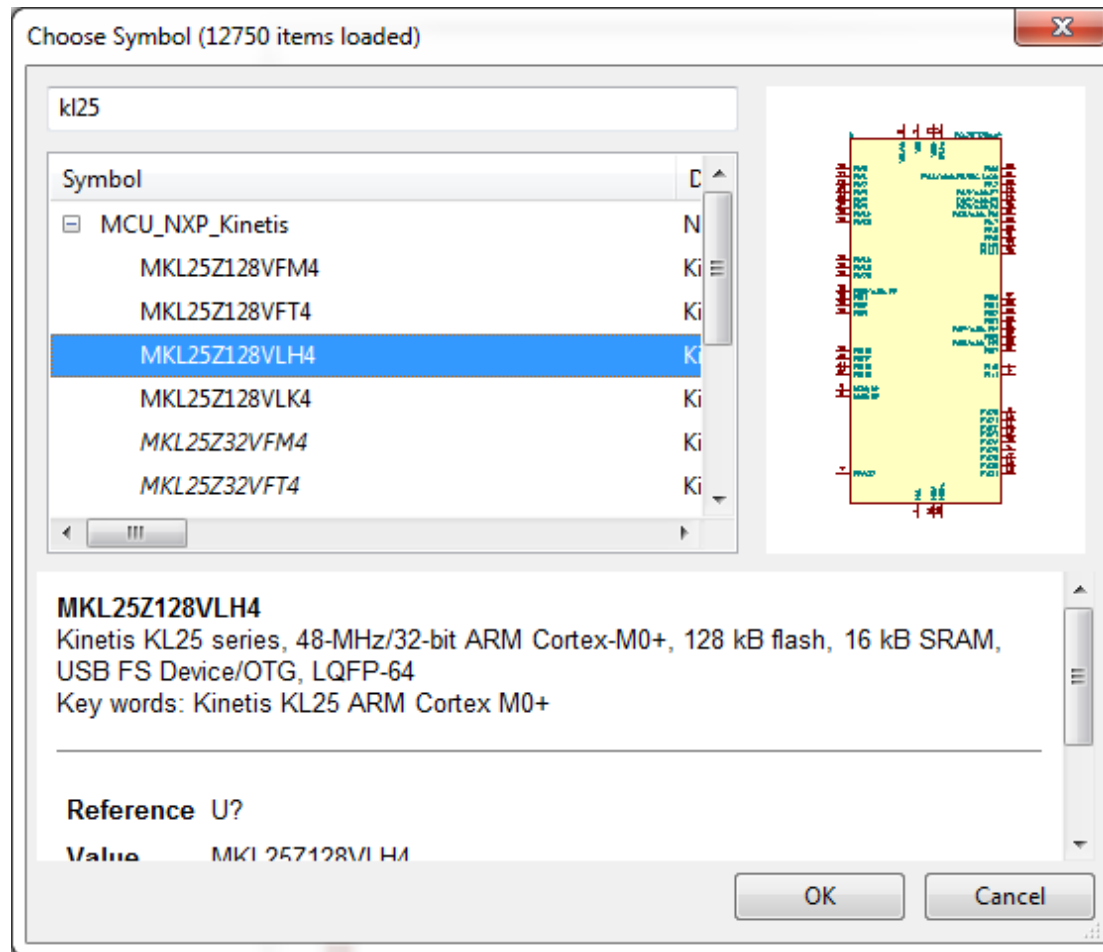
OK Cancel

Comment 4 on Top

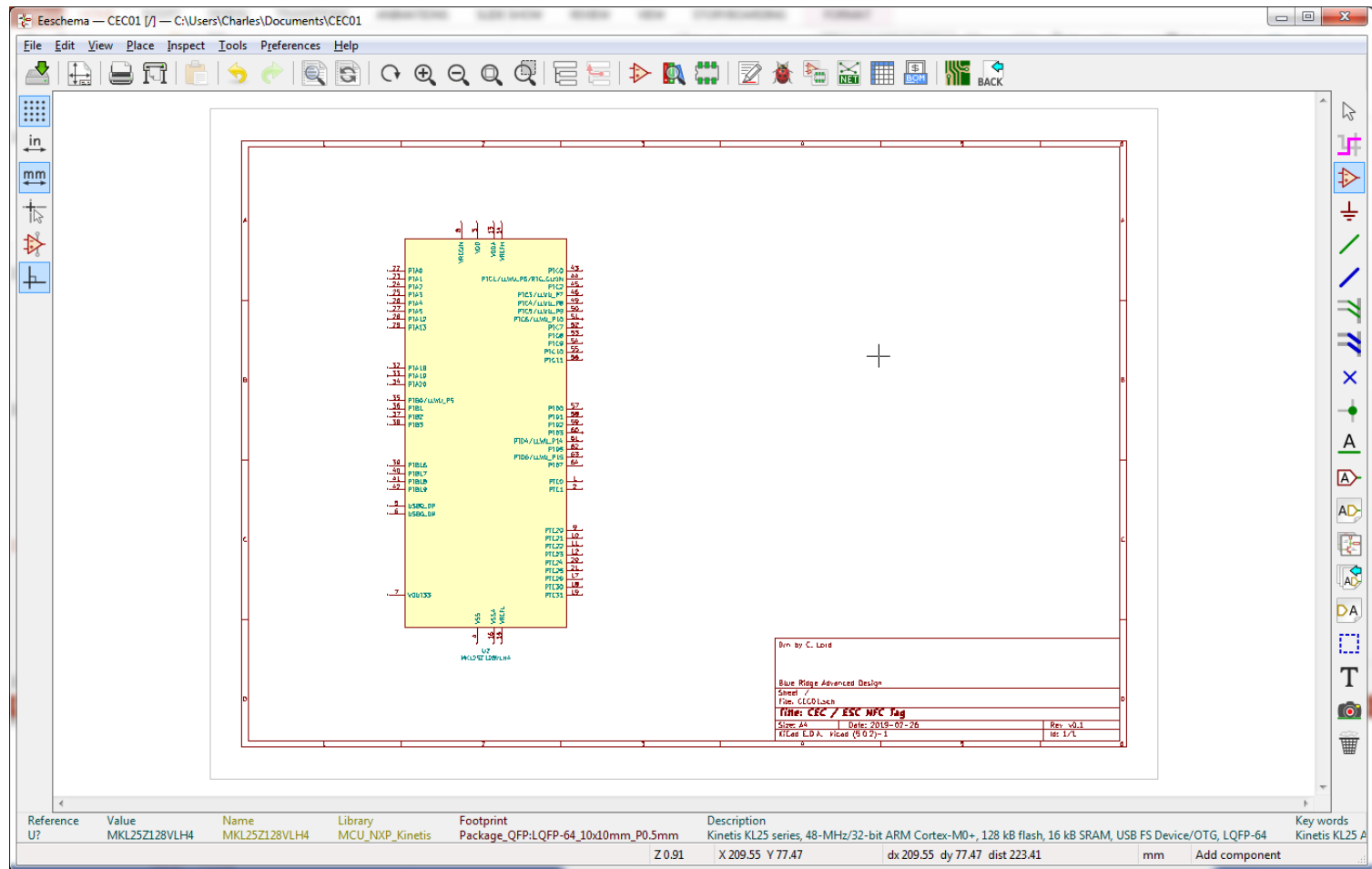


Presented by:

Built-in Library has our Chip

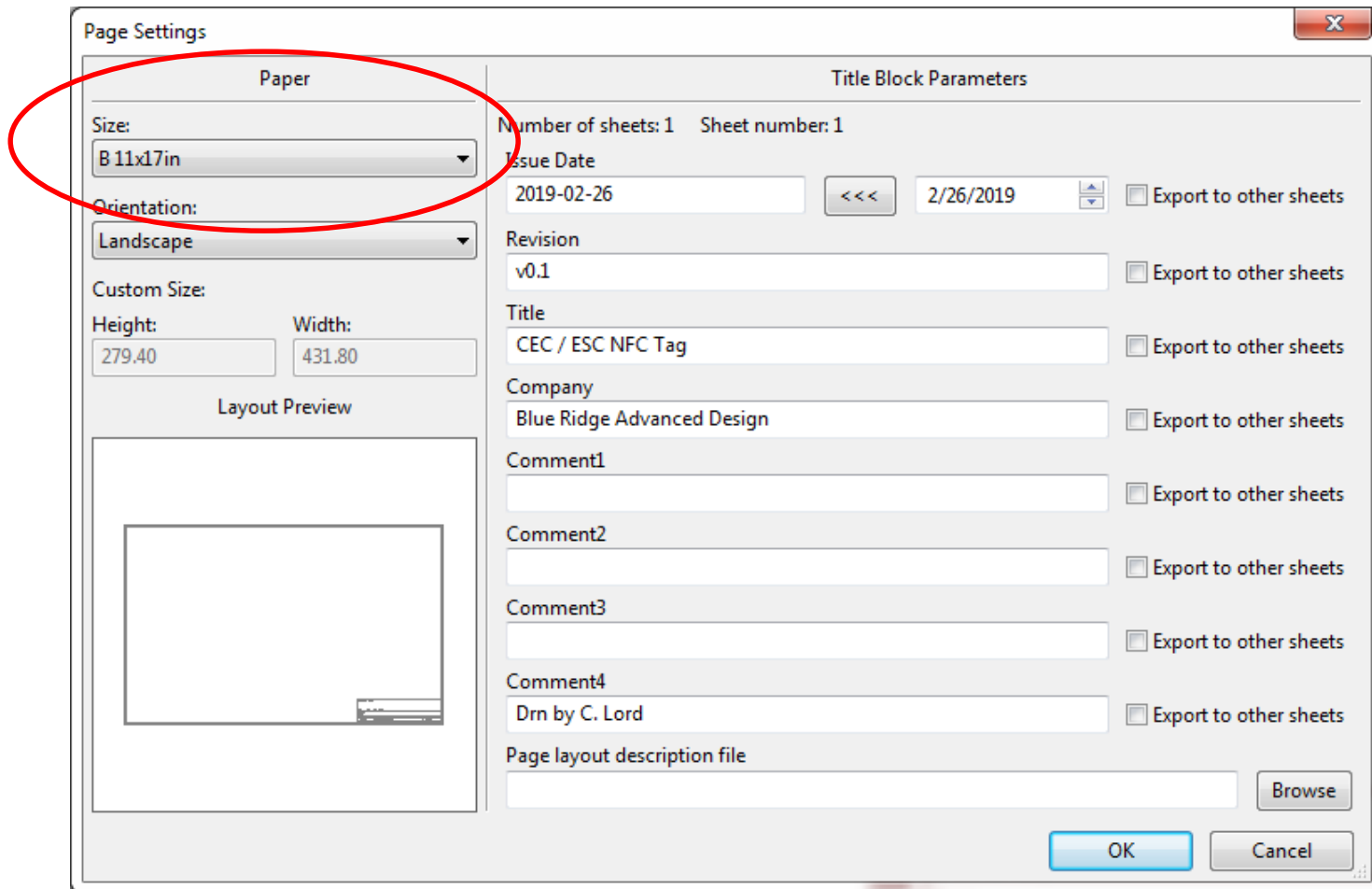


There is our Microcontroller

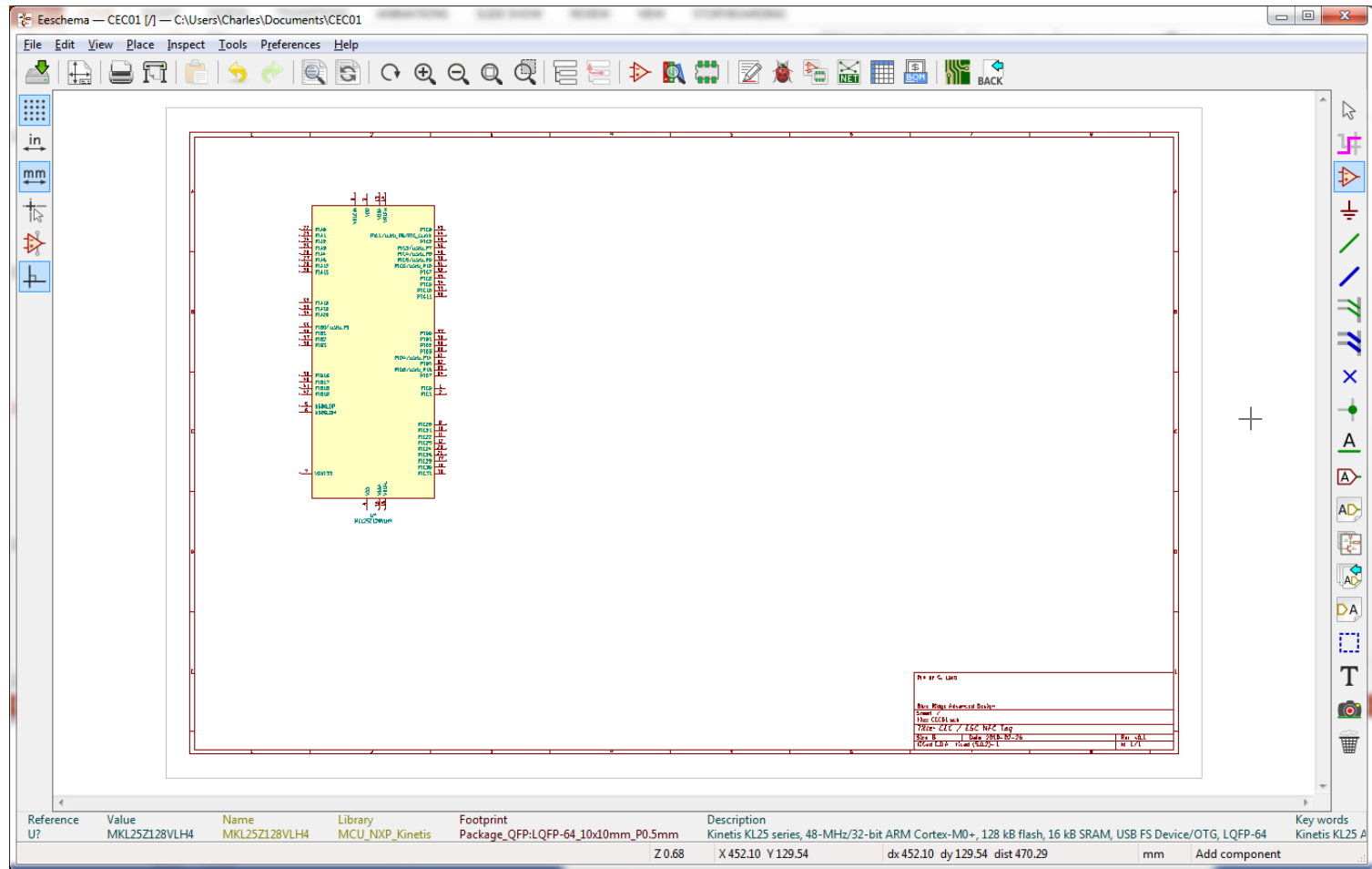


Presented by:

Change to B (11x17)

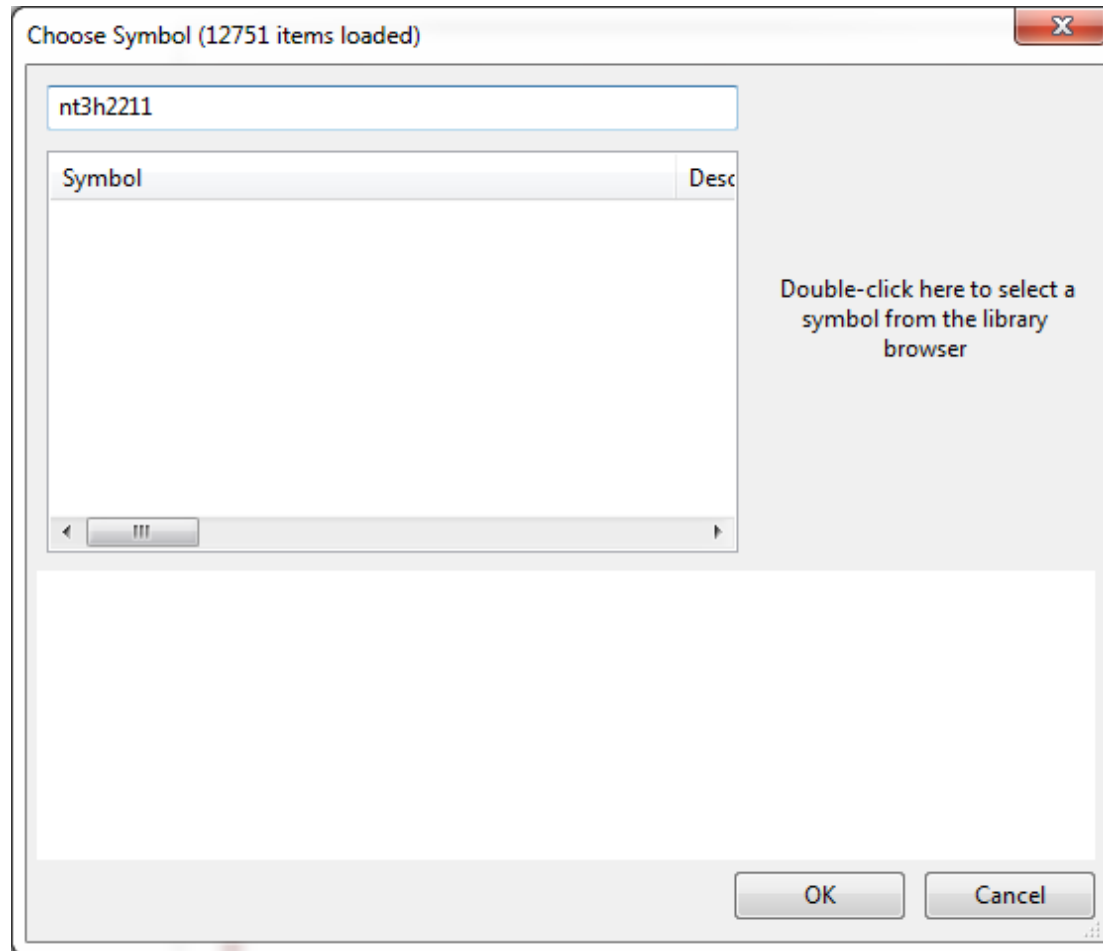


Now we have room



Presented by:

Now for the NTAG



Let's Create it!

Symbol Properties

General Settings:

Symbol name: NT3H2211

Default reference designator: U

Number of units per package: 1

Create symbol with alternate body style (DeMorgan)

Create symbol as power symbol

Units are not interchangeable

Pin Settings:

Pin text position offset: 40

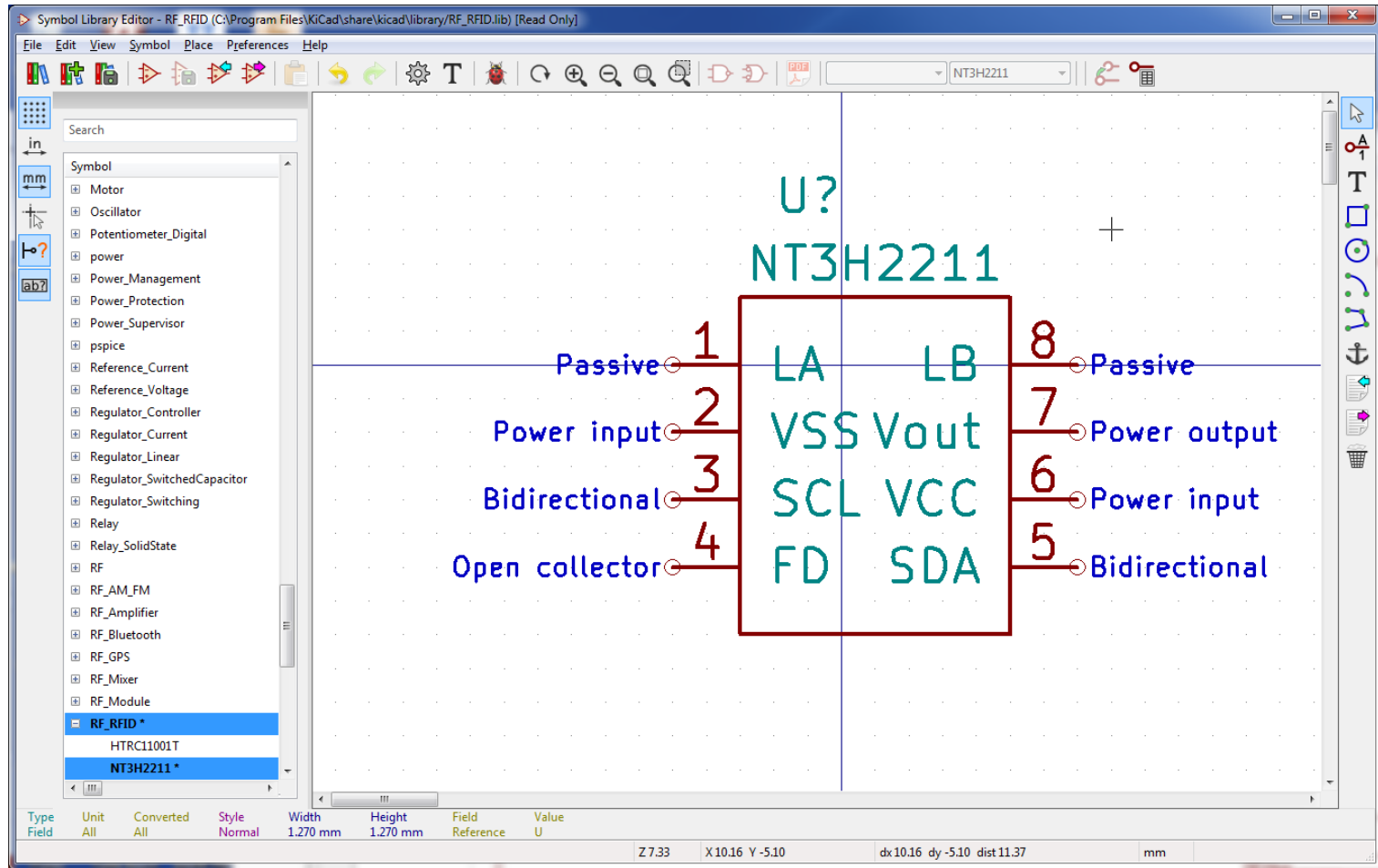
Show pin number text

Show pin name text

Pin name inside

OK Cancel

Define 8 pins, draw rectangle

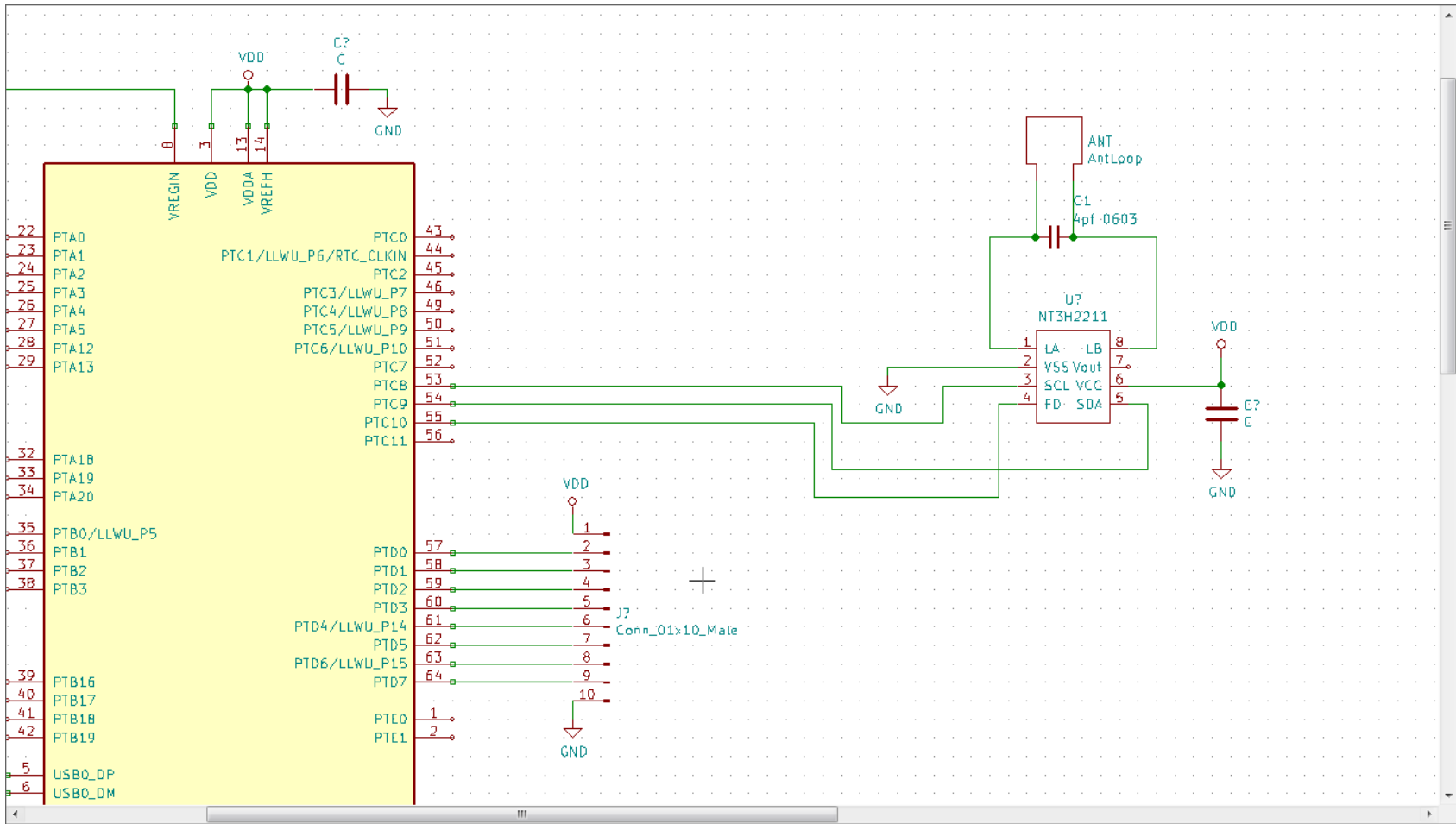


Presented by:

We Need a I²C Port

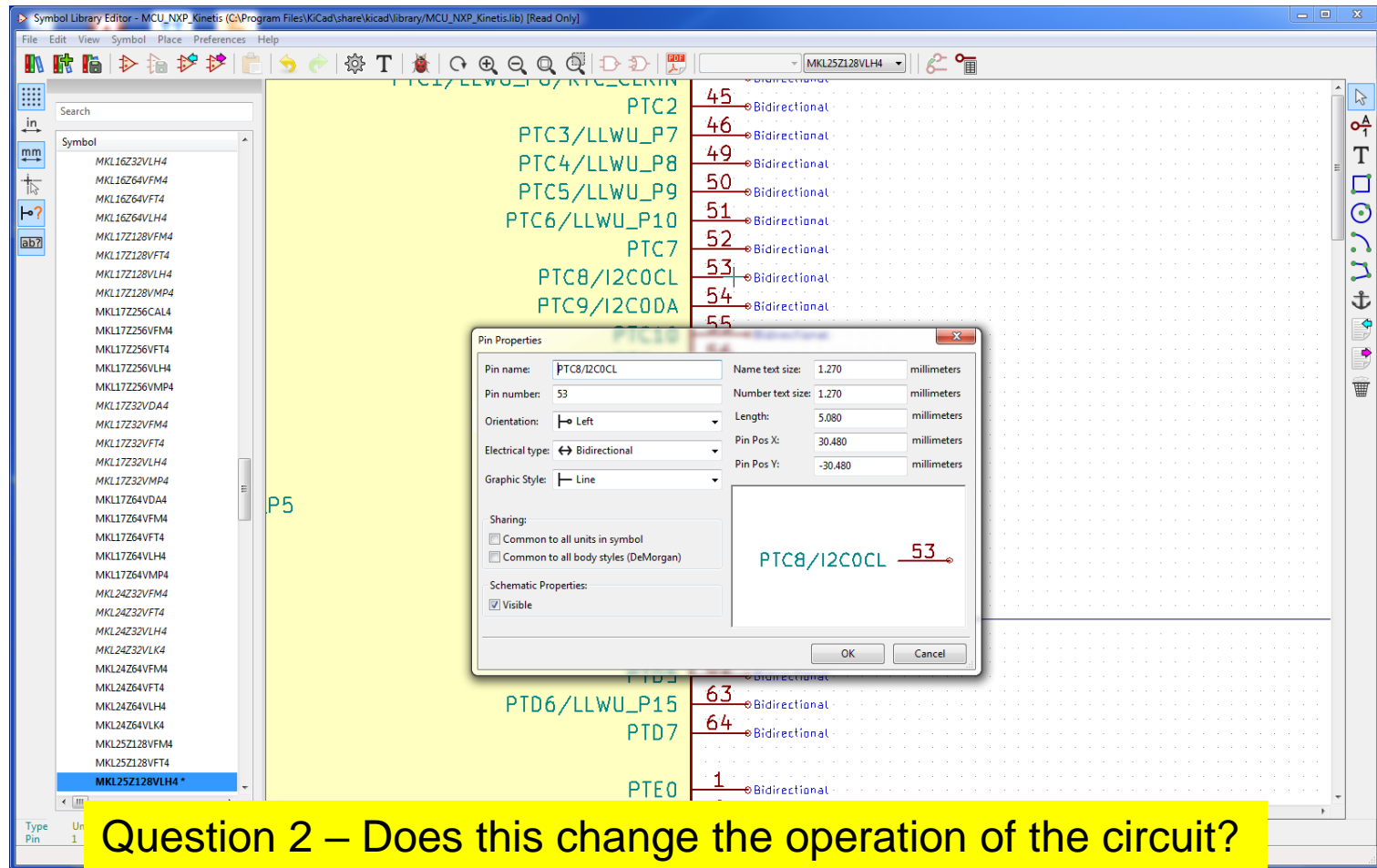
				LLWU_P10			LLWU_P10	
64	52	40	28	PTC7	CMP0_IN1	CMP0_IN1	PTC7	SPI0_MISC
65	53	—	—	PTC8	CMP0_IN2	CMP0_IN2	PTC8	I2C0_SCL
66	54	—	—	PTC9	CMP0_IN3	CMP0_IN3	PTC9	I2C0_SDA
67	55	—	—	PTC10	DISABLED		PTC10	I2C1_SCL
68	56	—	—	PTC11	DISABLED		PTC11	I2C1_SDA
69	—	—	—	PTC12	DISABLED		PTC12	

PTC8 and PTC9 are Mis-labeled

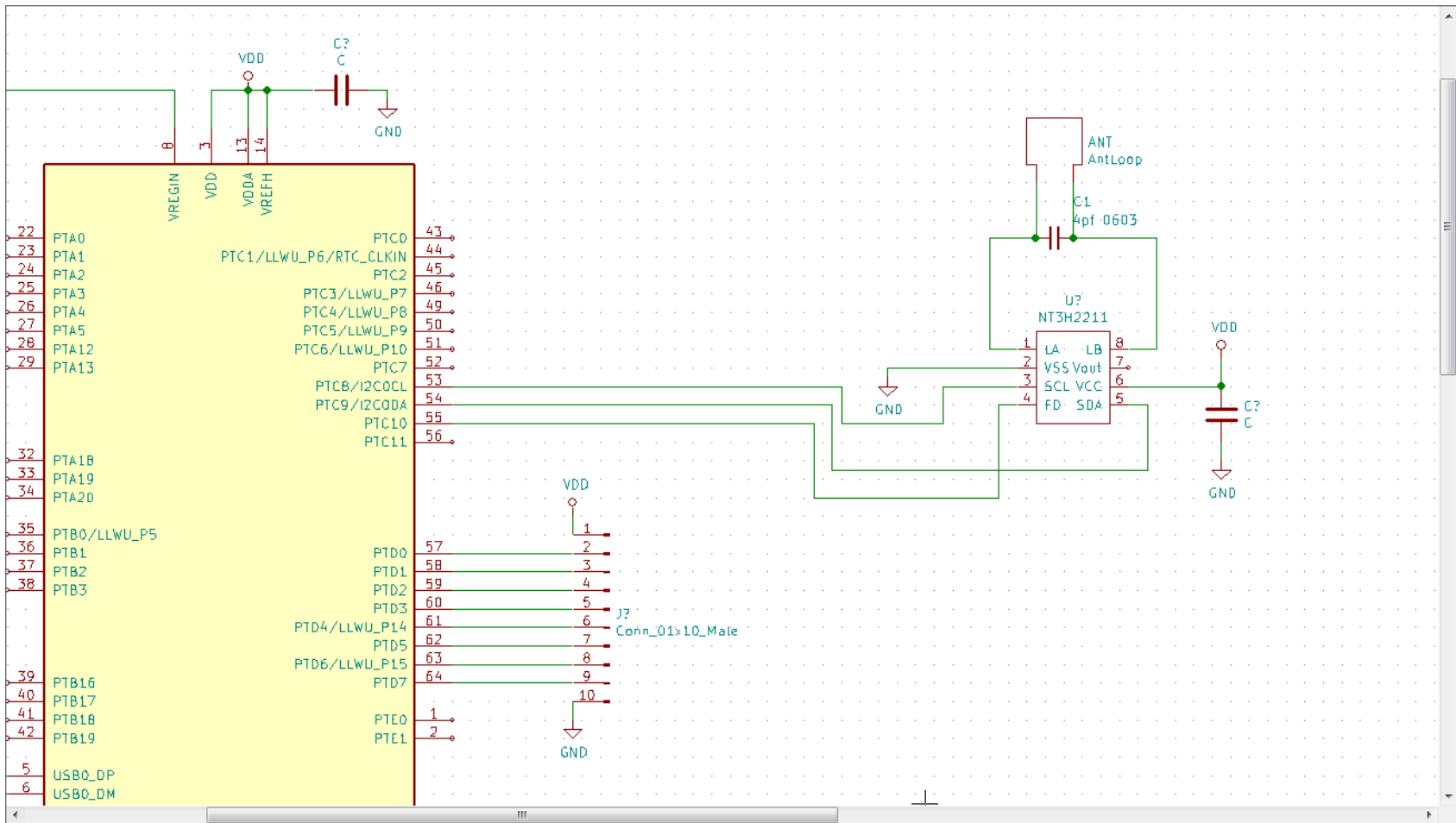


Presented by:

We add the functions to pin description



Saved changed KL25 to our library



Presented by:

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



TC2030-CTX Plug-of-Nails™ Cable

ARM Cortex 10-pin Cable with 6-Pin TC2030 Plug-of-Nails™ Connector.



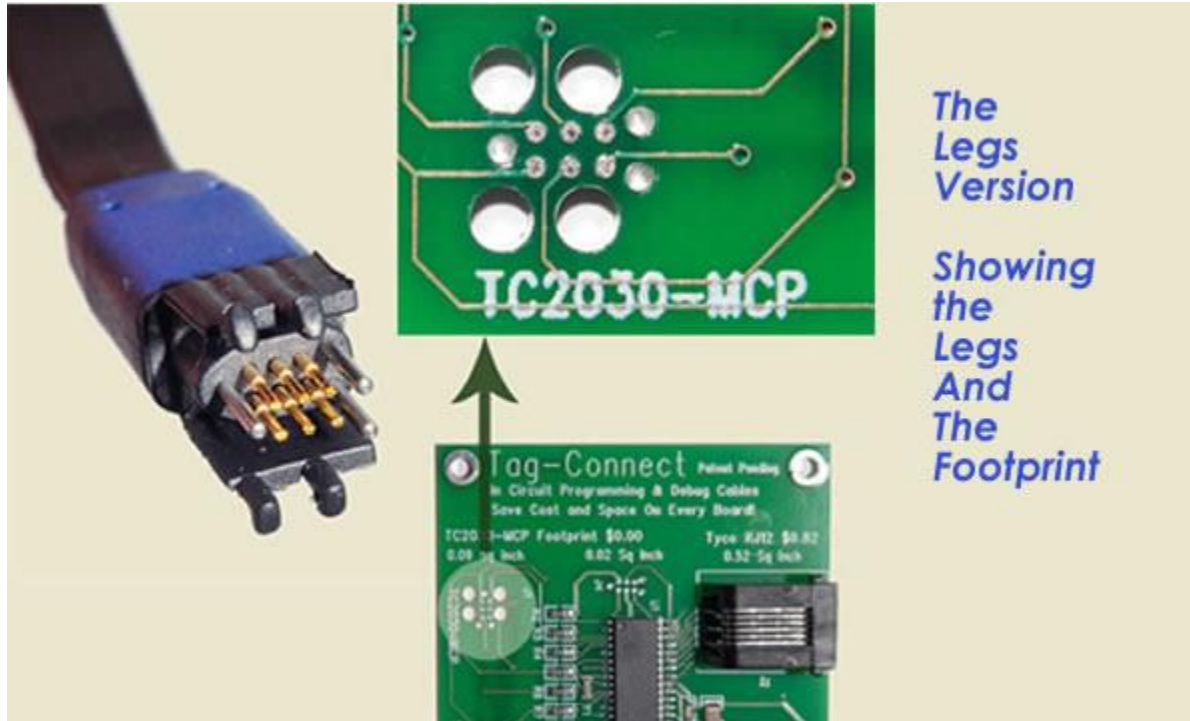
Tag-Connect Plug-of-Nails™
TC2030-NL
0.02 Sq Inch
0.06 Sq Inch
Cortex SWD
FTSH-105 micro header

Footprint Space Comparison
66% Space Saving

ARM 20-Pin
TC2030
ARM 20-Pin vs TC2030
84% Space Saving

(Illustration only - ULINK2 not included!)

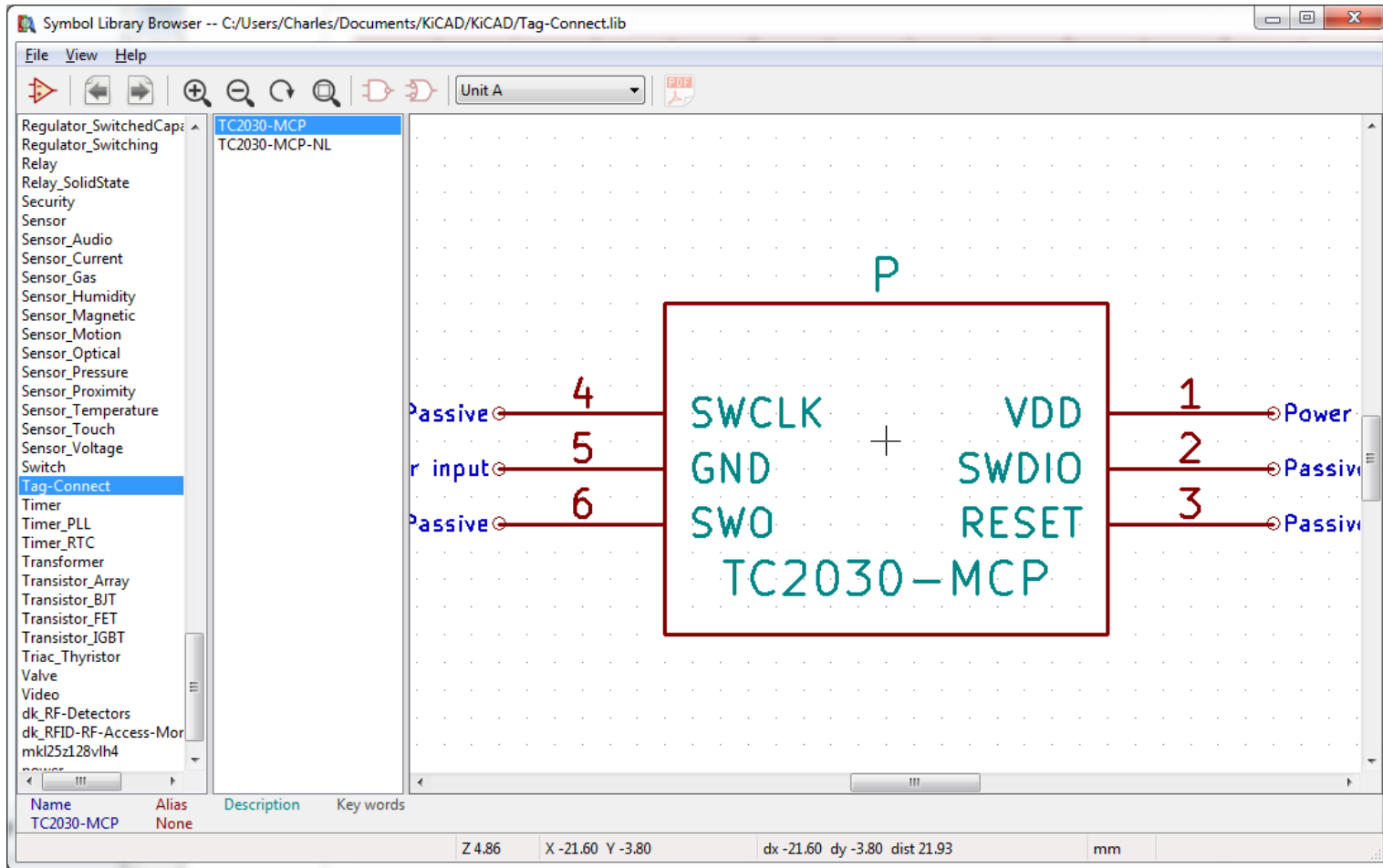
The Important Info



6-Pin TC2030 Footprint

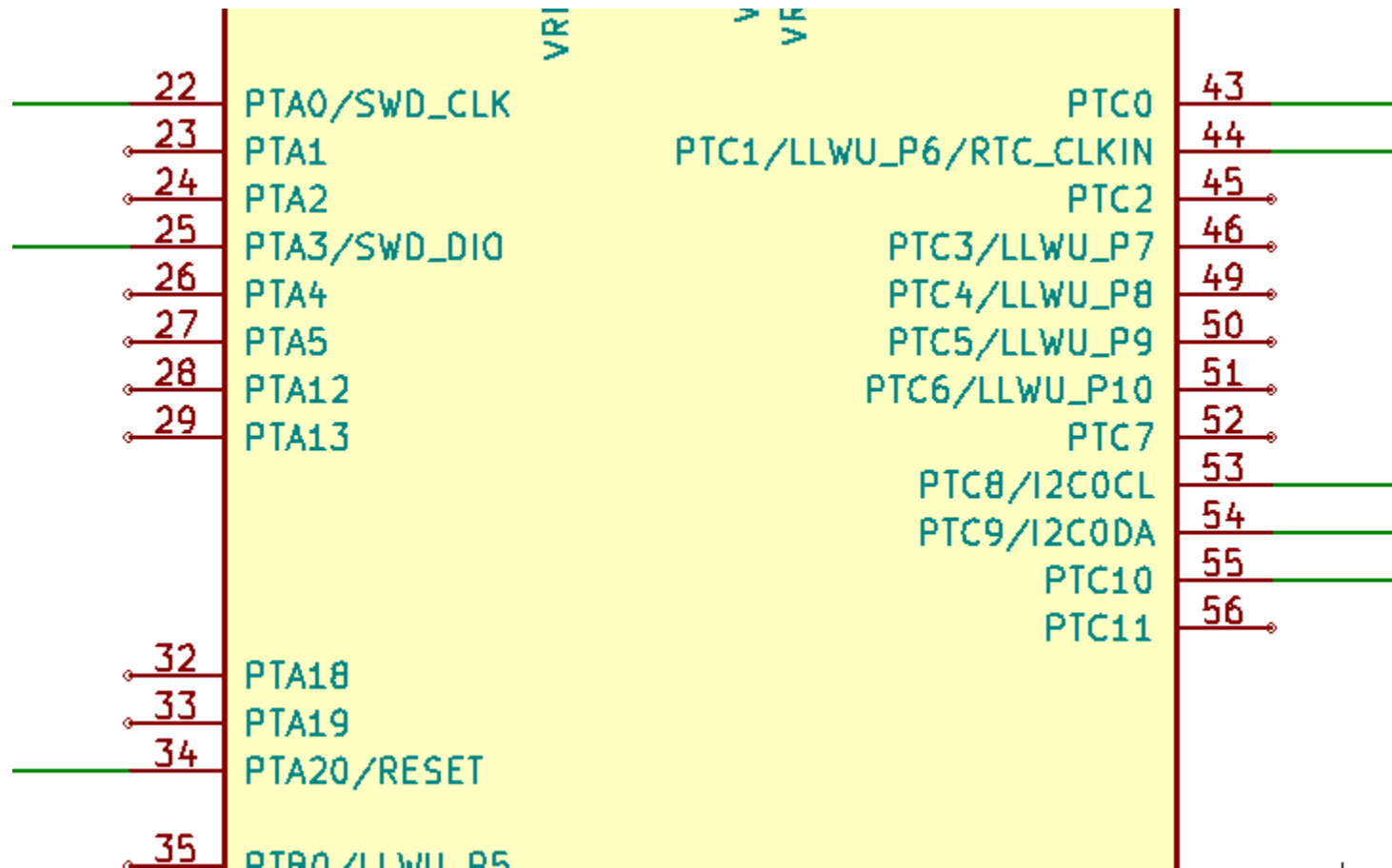
- 1 VCC
- 2 SWDIO / TMS
- 3 nRESET
- 4 SWCLK / TCK
- 5 GND (also connected to GNDDetect)
- 6 SWO / TDO

Symbol

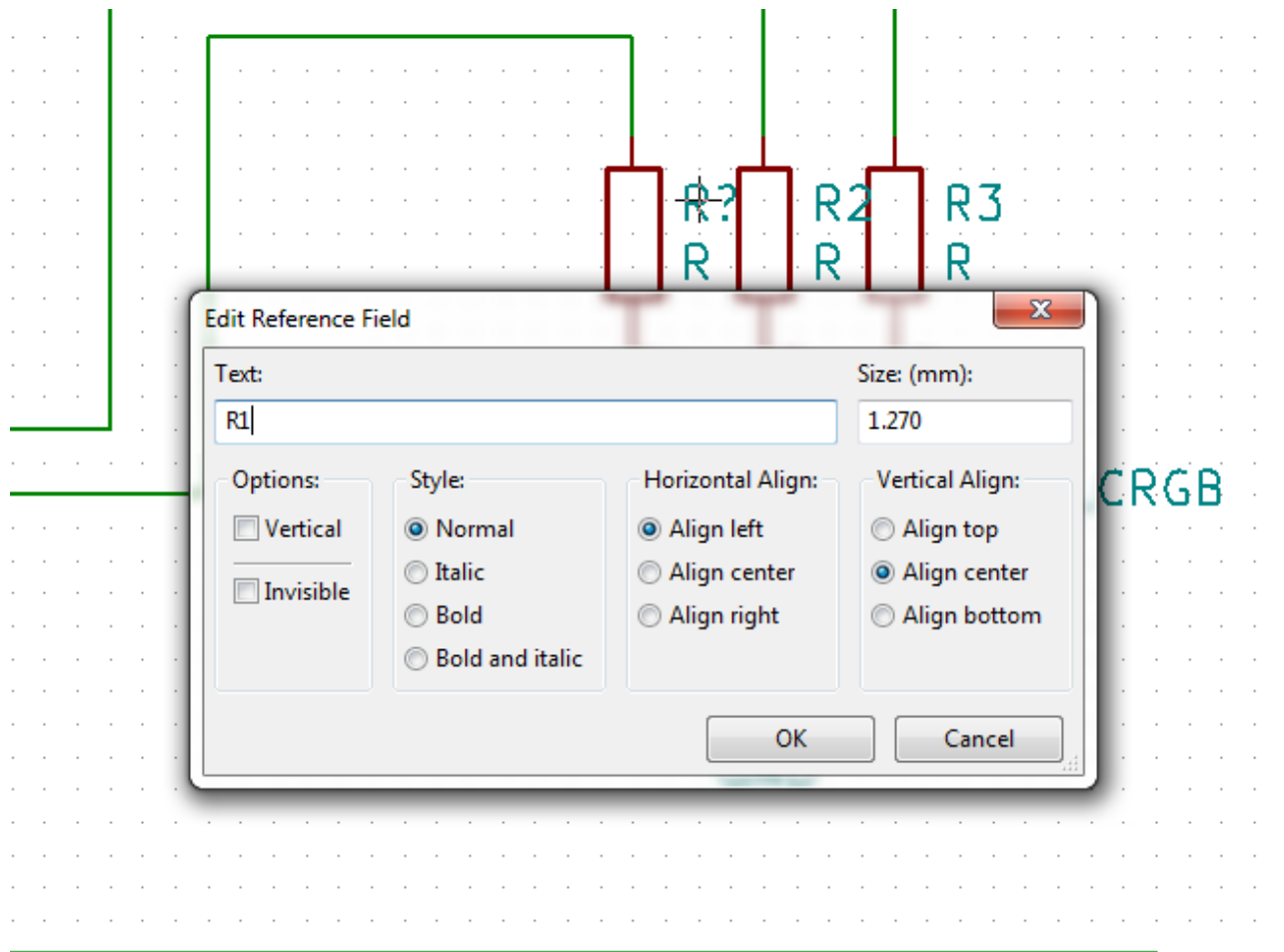


Presented by:

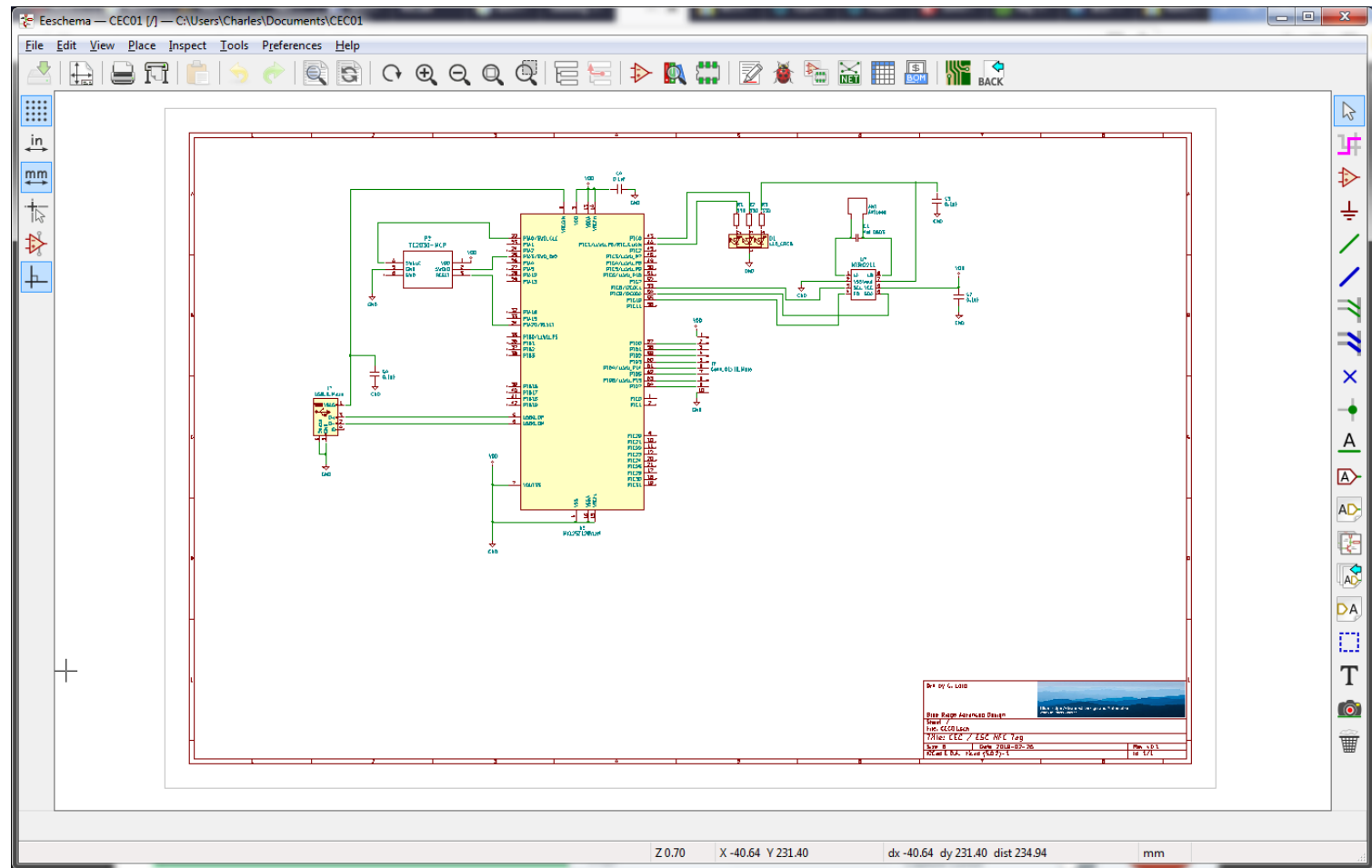
Added SWD Labels



Generic R and C, Edit Parameters

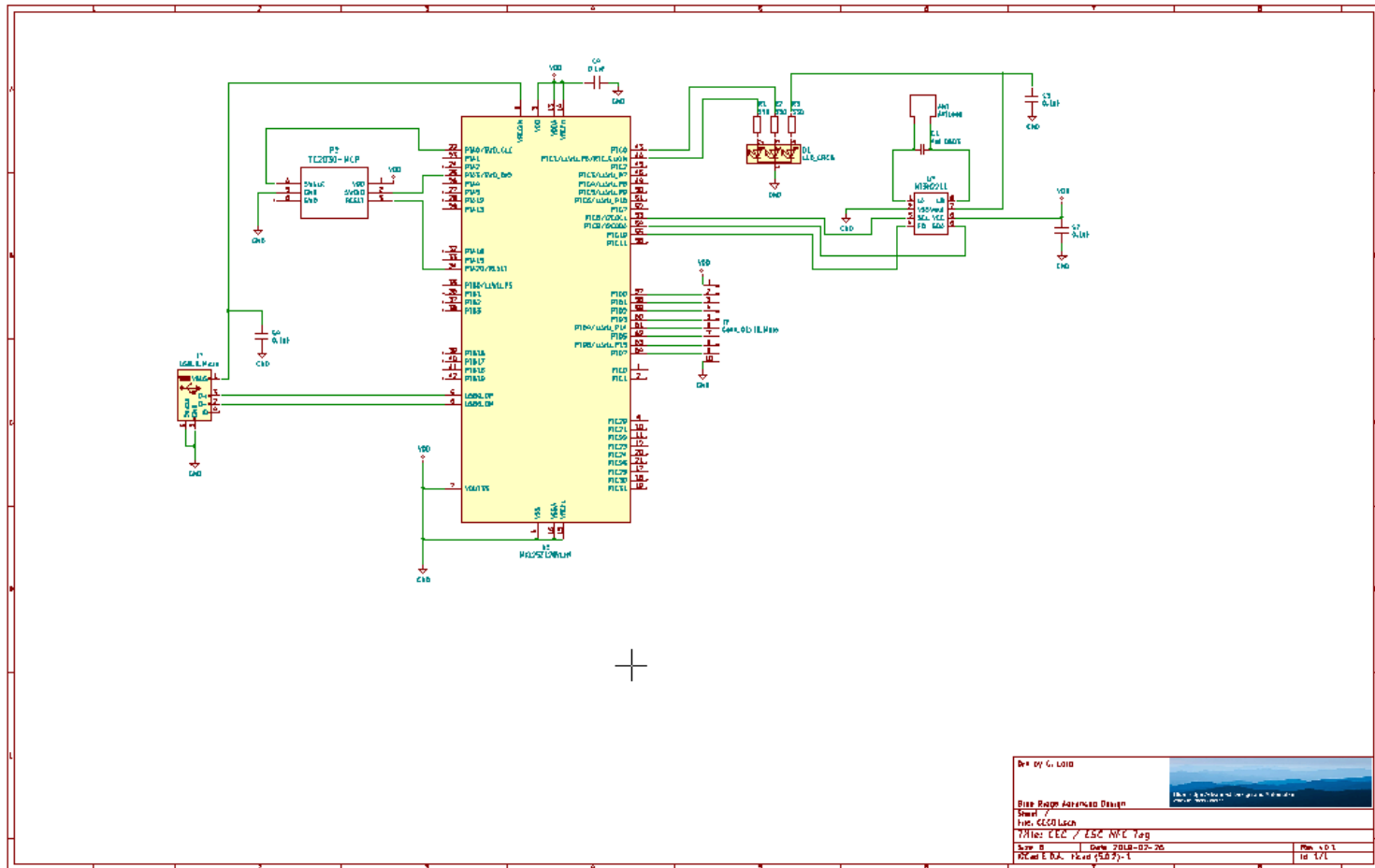


Added a bitmap (logo)



Presented by:

A little small on B paper



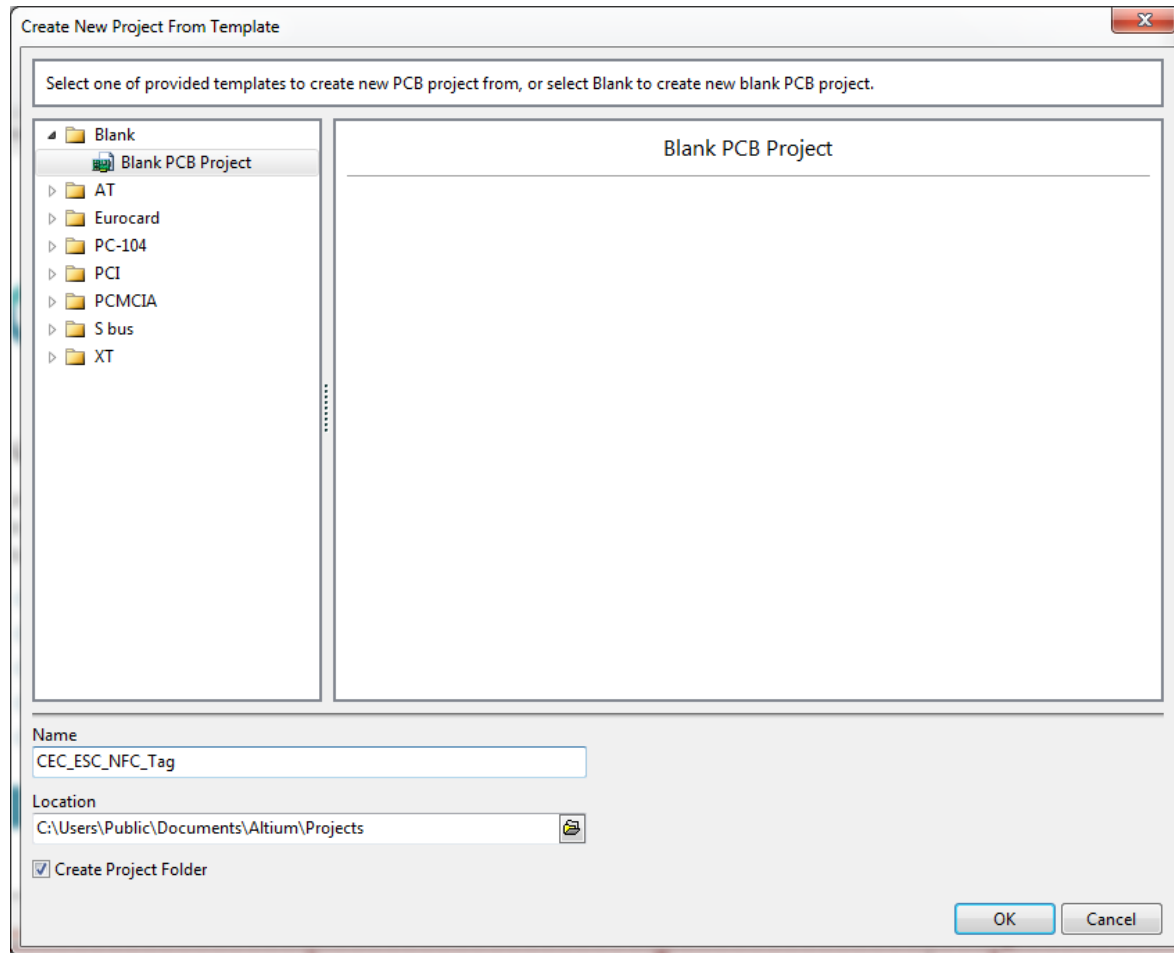
Des. by G. Loto

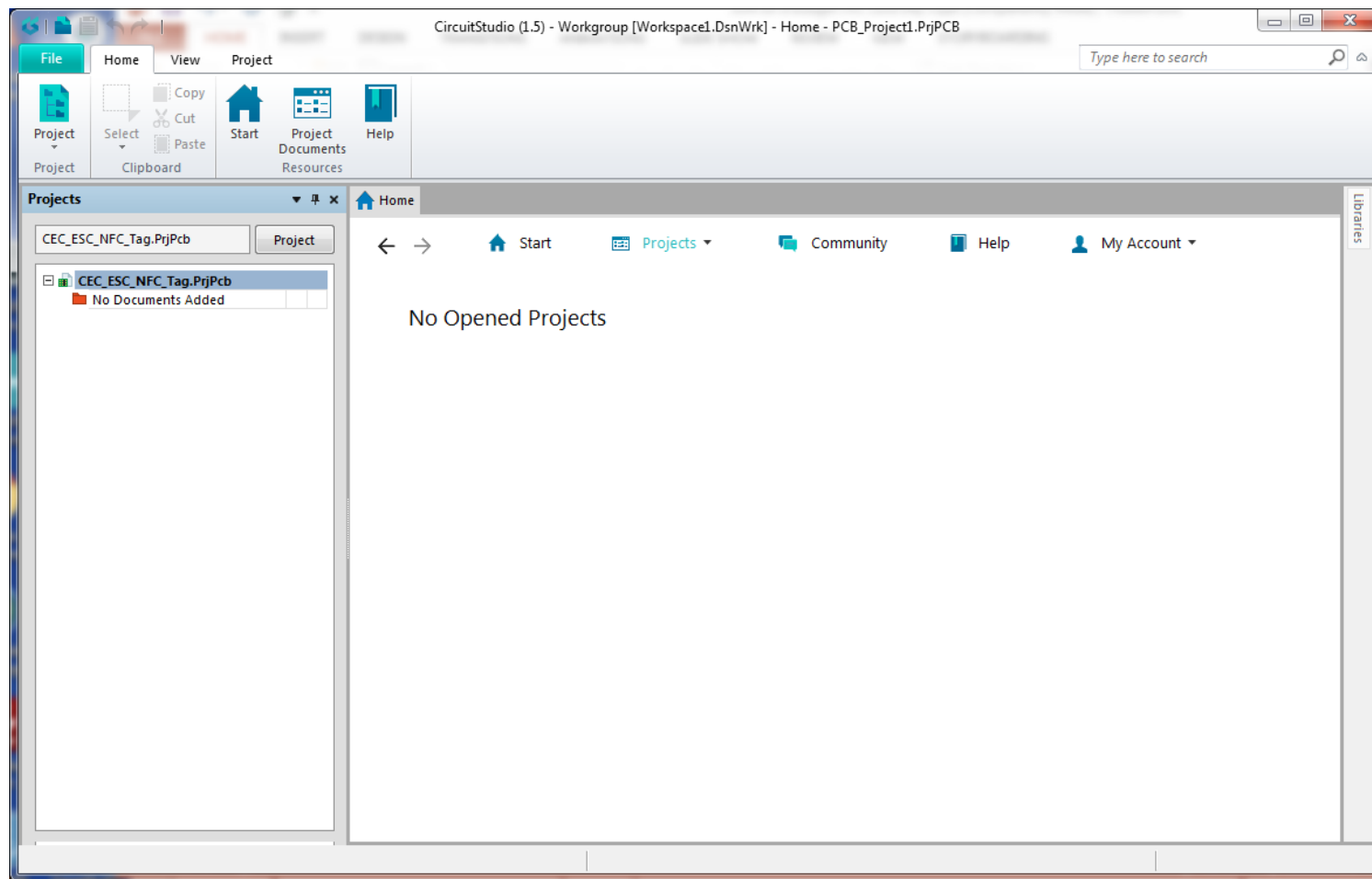
Blue Ridge Advanced Design
Street /
P.O. Box 1000
TVA: CEC / ISE, N.E. 70g
Rev. 0 Date: 2018-03-26 Pch. 4 of 1
PCB: E. B.A. 16x24 (A2) 1 to 1/1

CircuitStudio

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

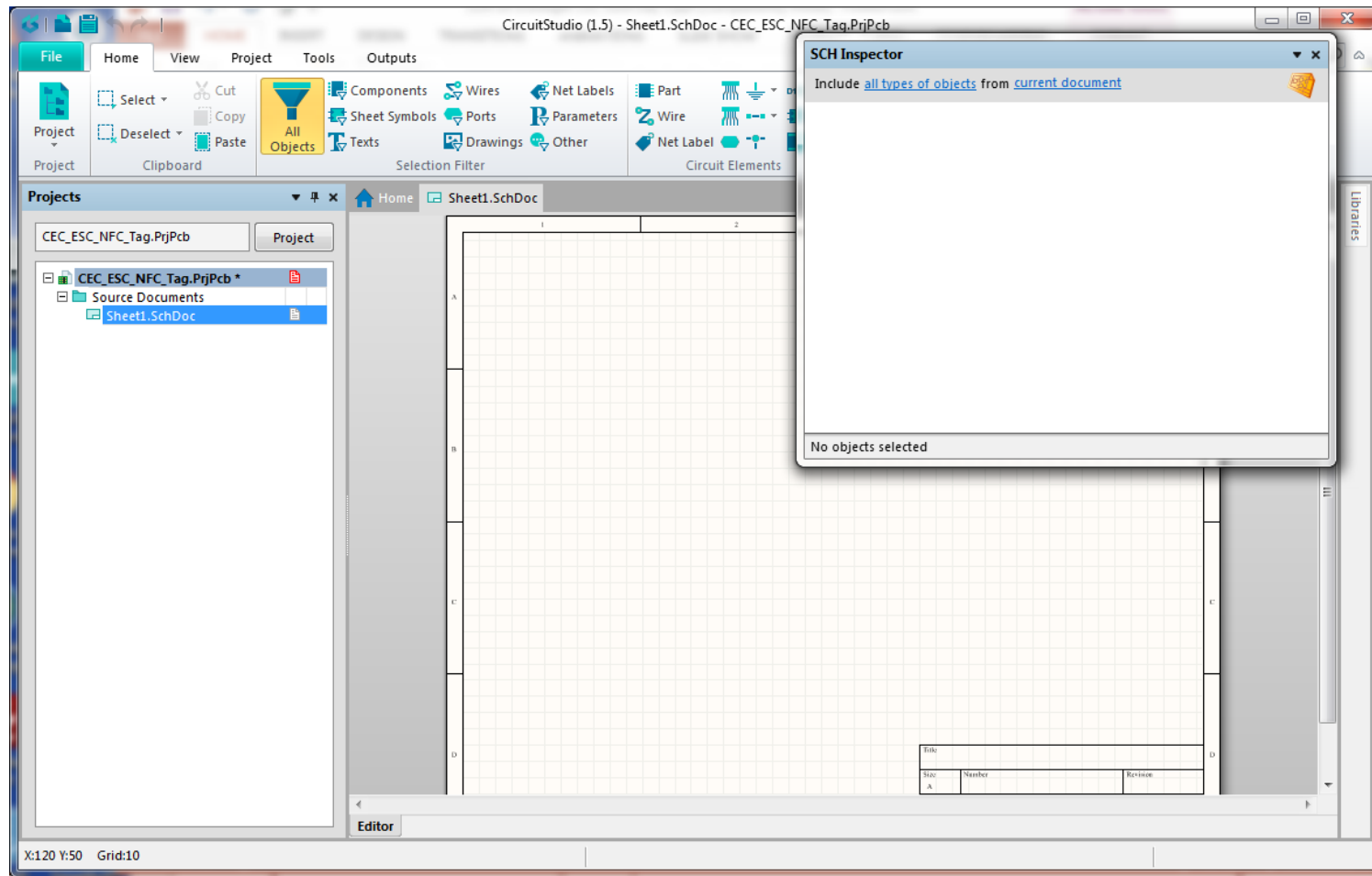
Click on File, New PCB Project





Presented by:

Project, Add New Schematic



Presented by:

Vault Explorer, KL25Z?

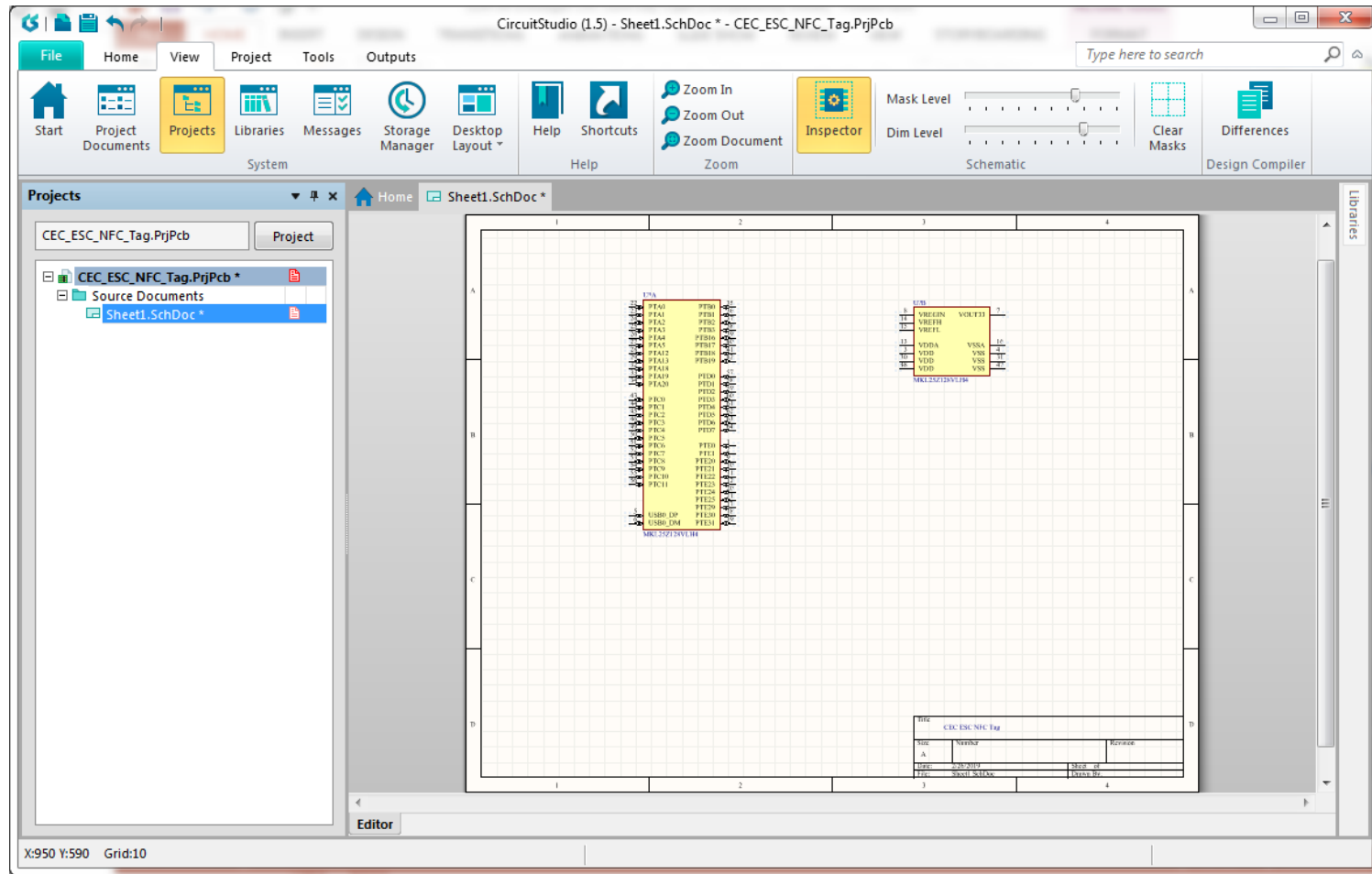
The screenshot shows the Vault Explorer application window. The left pane displays a tree view of 'Vault Folders' under 'Altium Content Vault', including various manufacturers like Crystek, CSR, CTS Corporation, etc. The right pane shows a search result for 'MKL25Z128VLH4' under the category 'No Category'. The component is listed in a table with columns for Item, Revision, State, Description, Comment, and Note.

Item	Revision	State	Description	Comment	Note
⊕ CMP-1750-00001	1	Released	Kinetis K22F Sub-Family, 120 ...	MK22FN1M0...	
⊖ CMP-1750-00002	1	Released	48 MHz Cortex-M0+ Based Mi...	MKL25Z128V...	
⊕ CMP-1750-...	1	Released	48 MHz Cortex-M0+ Based Mi...	MKL25Z128V...	
⊕ CMP-1750-00003	1	Released	3-Axis, 12-Bit / 8-Bit Digital Ac...	MMA8452QR1	
⊕ CMP-1750-00004	1	Released	Proximity Capacitive Touch Se...	MPR121QR2	

Below the table, a detailed view of the selected component is shown: **MKL25Z128VLH4 [CMP-1750-00002-1] Released**. The description reads: "48 MHz Cortex-M0+ Based Microcontroller with USB, 1.71 to 3.6 V , -40".

Presented by:

Part A and B



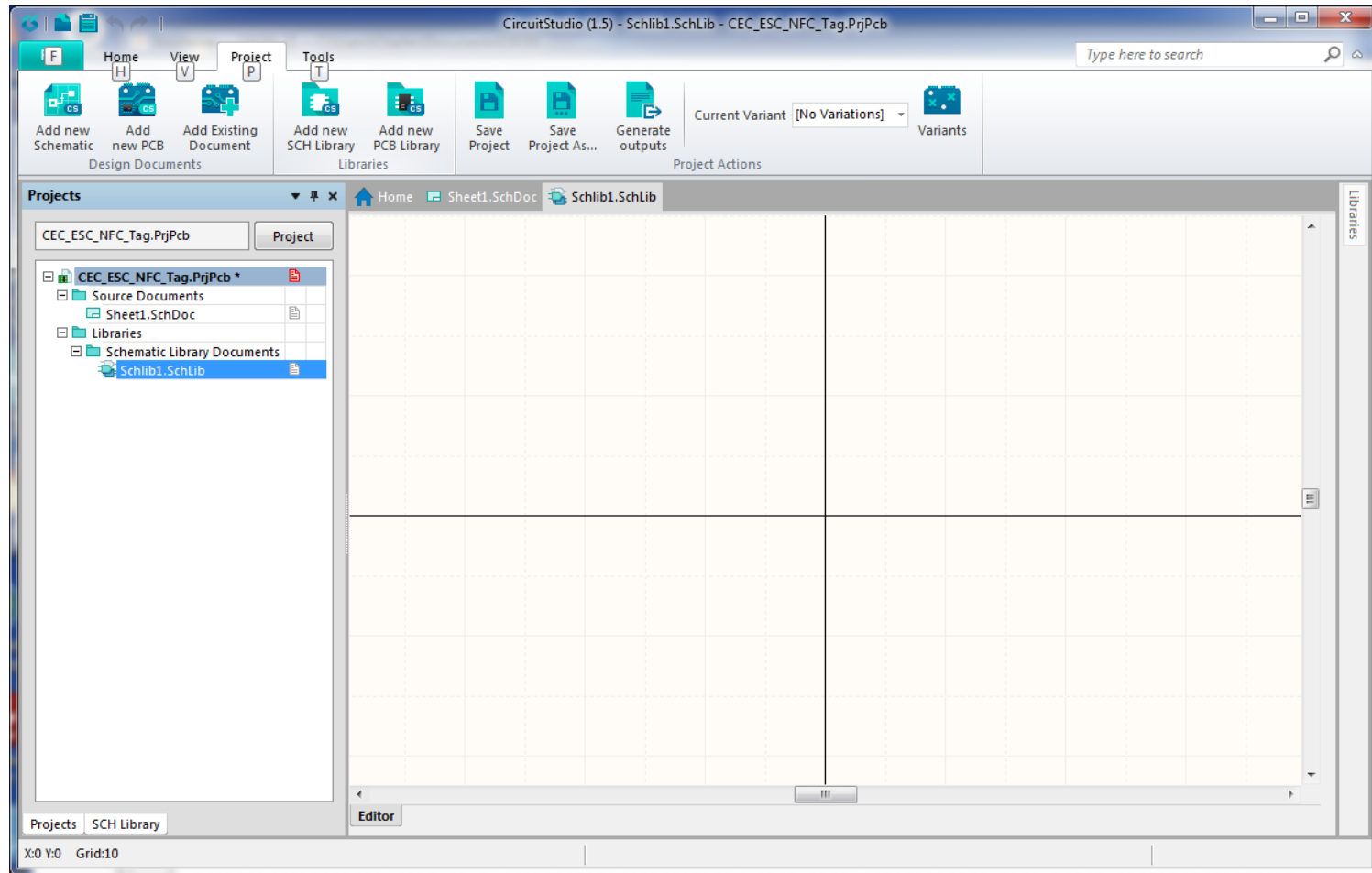
Presented by:

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:

Comment	
<input type="text" value="NT3H1101W0FHKH"/>	
Description	
<input type="text" value="NTAG I2C-Energy Harvesting NFC Forum Type 2 Tag with Field Detection Pin and I2C Interface, 1.7 to 3.6 V, -40 to 95 degC, 8-Pin QFN (SOT-902-3), RoHS, Tape and Reel"/>	
Folder	Ancestor Revision
<input type="text" value="Unified Components\Components\New Releases\New"/>	<input type="text"/>

We Need to Create



Presented by:

CircuitStudio (1.5) - Schlib1.SchLib * - CEC_ESC_NFC_Tag.PrjPcb

Home View Project Tools

Add new Schematic Add new PCB Add Existing Document Add new SCH Library Add new PCB Library Save Project Save Project As... Generate outputs

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 *

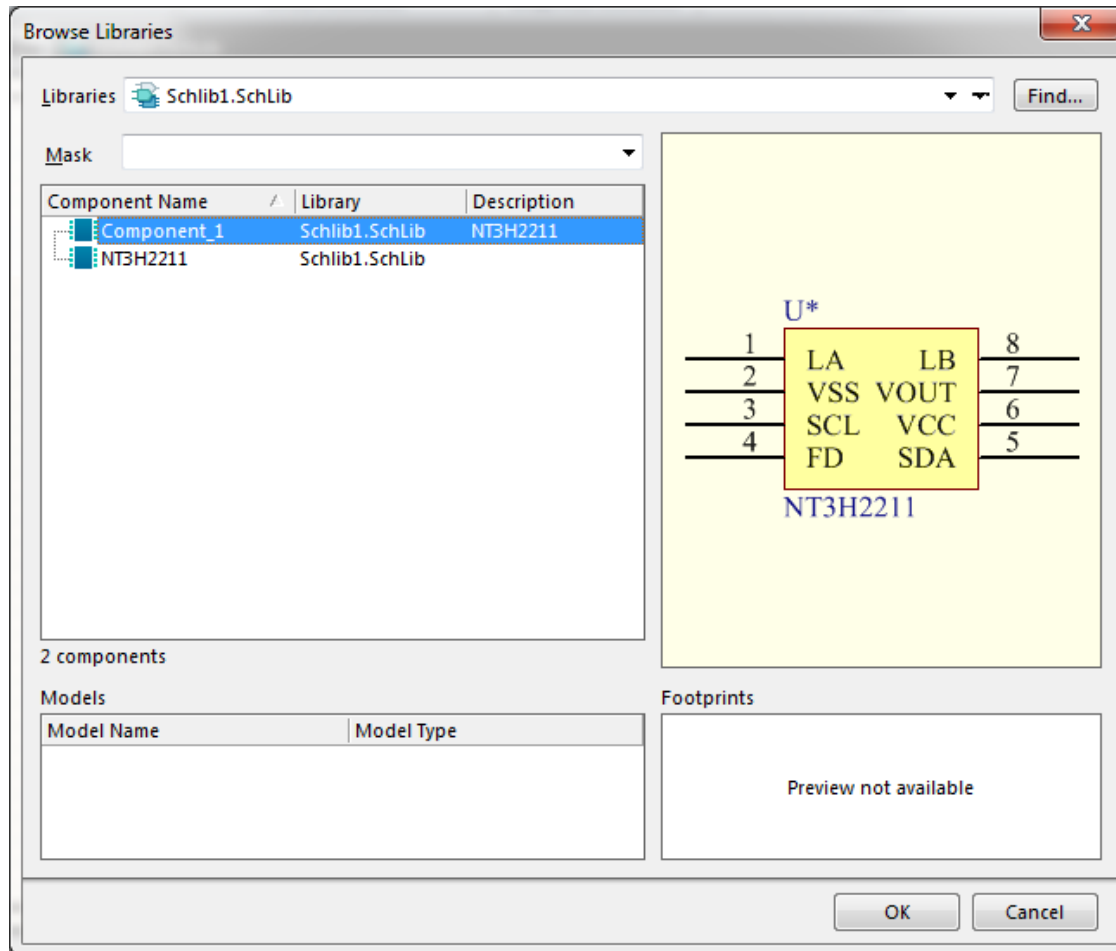
Home Sheet1.SchDoc Schlib1.SchLib *

1 2 3 4 | 1 2 3 4 | 8 7 6 5

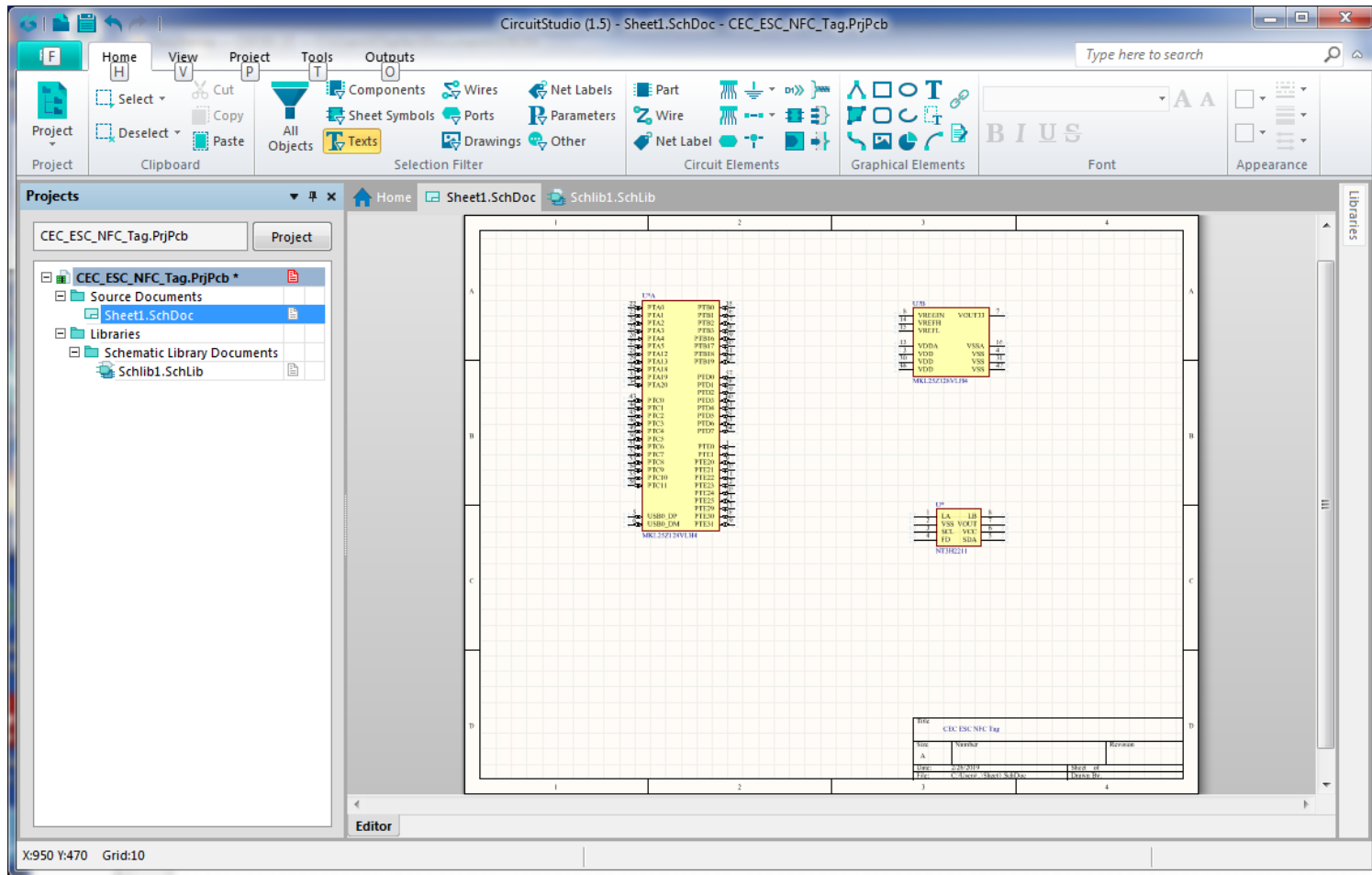
Editor

X:80 Y:50 Grid:10 Hit Spacebar to change mode

Finished Part



IC's In Place



Presented by:

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/bradatrainning>