Electrical and Computer Engineering

ECE315

Lab 2:

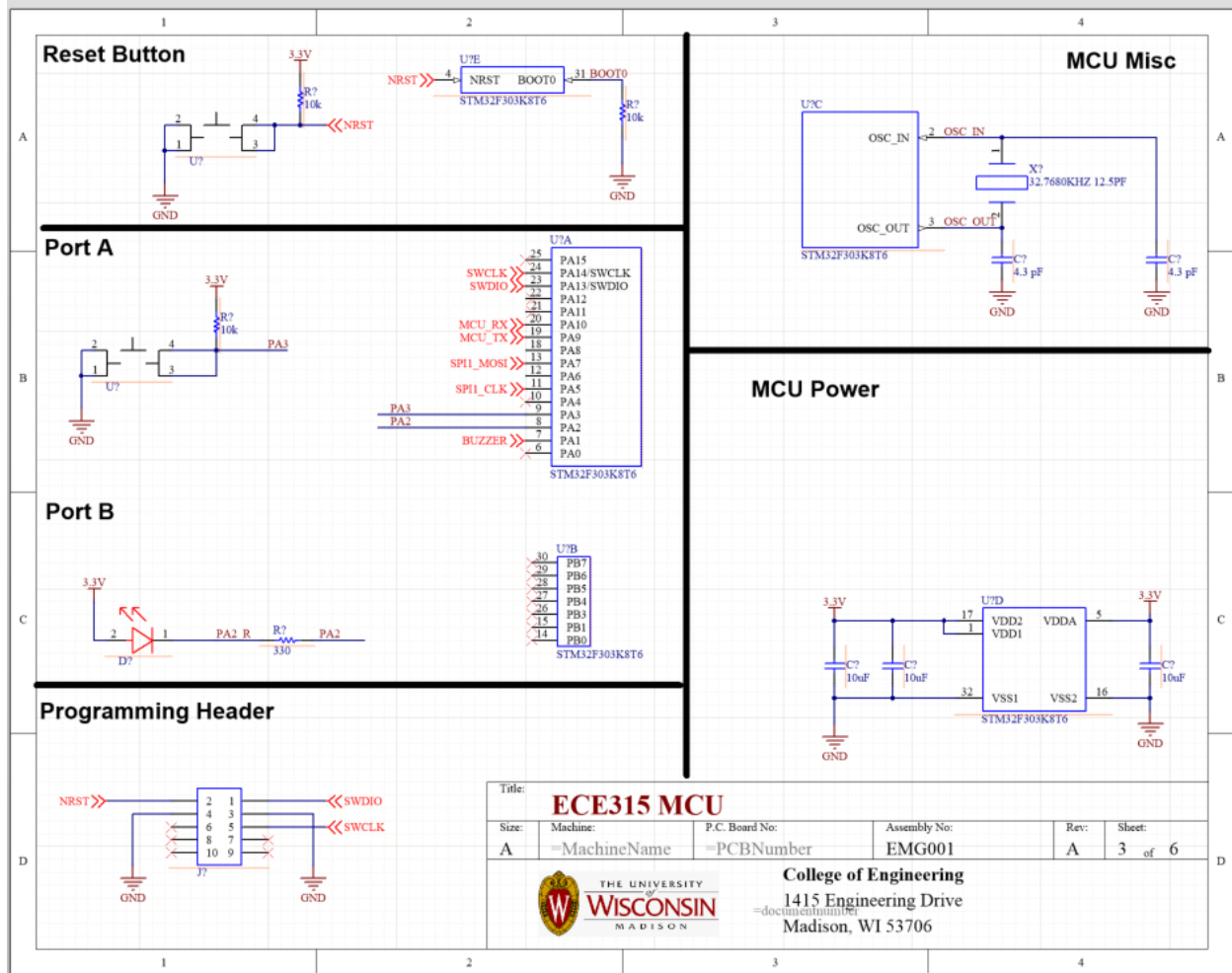# Schematic Capture

1. Lab 2 Overview

In Lab 1, you created schematic symbols and PCB footprints for several components. In this lab, you will place and connect those components in a series of schematic sheets. Your schematic sheets represent the components found in your embedded system and how they are connected to one another.
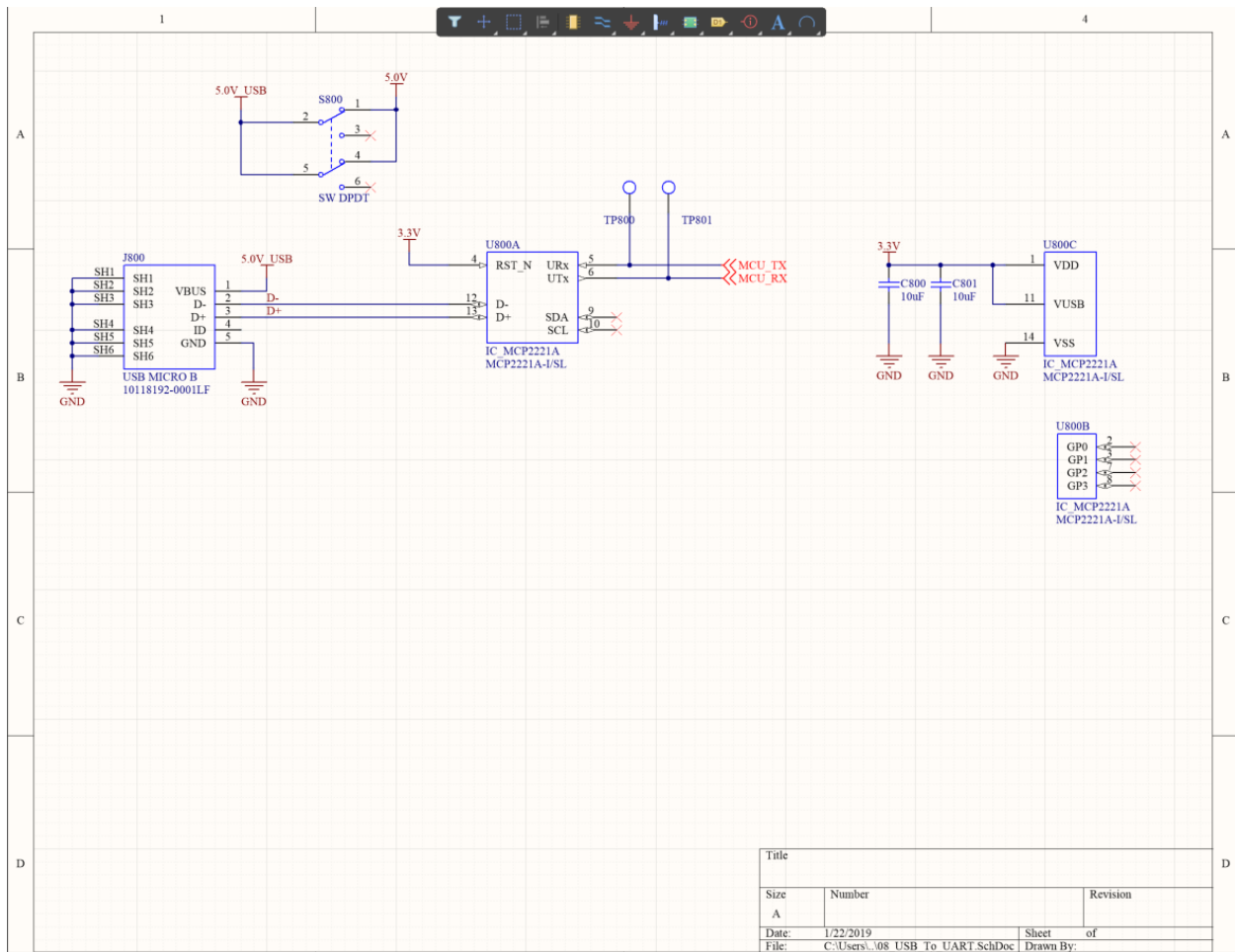
2. Pre-Lab

1. Open the Altium Designer Project that you completed in Lab 1 by clicking on the **ECE315.PrjPcb** PCB Project File.

2. Watch the following **videos** to complete 03-MCU.SchDoc and 08-USB_To_UART.SchDoc.

    Schematic Capture Basics
    Schematic Capture Tutorial

    Use the information in the videos to complete pages 3 and 8 of the schematics. Once you are finished, your schematics should look like the following images:

Reset Button

U?E
NRST >> 4 | NRST BOOT0 | 31 BOOT0
STM32F303K8T6

3.3V
R?
10k
<< NRST

R?
10k
GND

U?
GND

Port A

U?A
| 25 | PA15 |
| 24 | PA14/SWCLK | SWCLK >>
| 23 | PA13/SWDIO | SWDIO >>
| 22 | PA12 |
| 21 | PA11 |
| 20 | PA10 | MCU_RX >>
| 19 | PA9 | MCU_TX >>
| 18 | PA8 |
| 13 | PA7 | SPI1_MOSI >>
| 12 | PA6 |
| 11 | PA5 | SPI1_CLK >>
| 10 | PA4 |
| 9 | PA3 | PA3
| 8 | PA2 | PA2
| 7 | PA1 | BUZZER >>
| 6 | PA0 |
STM32F303K8T6

3.3V
R?
10k
PA3

U?
GND

Port B

U?B
30	PB7
29	PB6
28	PB5
27	PB4
26	PB3
15	PB1
14	PB0
STM32F303K8T6

3.3V
PA2_R R? PA2
330
D?

MCU Misc

U?C
OSC_IN 2 OSC_IN
X?
32.7680KHZ 12.5PF
OSC_OUT 3 OSC_OUT
STM32F303K8T6
C?
4.3 pF
GND
C?
4.3 pF
GND

MCU Power

3.3V
U?D
| 17 | VDD2 | VDDA | 5 | 3.3V
| 1 | VDD1 |
| 32 | VSS1 | VSS2 | 16 |
STM32F303K8T6
C?
10uF
C?
10uF
GND
C?
10uF
GND

Programming Header

NRST >>
2	1
4	3
6	5
8	7
10	9
J?
<< SWDIO
<< SWCLK
GND GND

Title: **ECE315 MCU**

| Size: | Machine: | P.C. Board No: | Assembly No: | Rev: | Sheet: |
| A | =MachineName | =PCBNumber | EMG001 | A | 3 of 6 |

THE UNIVERSITY of **WISCONSIN** MADISON

College of Engineering
1415 Engineering Drive
=documentnumber
Madison, WI 53706

3. Pre Lab: What to Turn in

File Name	Description
LastName_FirstName_Lab2_Altium.zip	Altium Project ZIP file. Delete the History folder before creating the ZIP file.
LastName_FirstName_Lab2_Sch.pdf	PDF of schematics with pages 03 and 08 completed. Use File→Smart PDF to generate the schematics. You can use the default settings, but only include the .SchDocs on the 3rd page of the dialog.

4. Lab Work

a. Complete 03_MCU.SchDoc

- Verify the following off sheet connectors are assigned to the GPIO pins listed below:

Signal Name	Connected to
SWDIO	PA13
SWCLK	PA14
BUZZER	PA1
MCU_RX	PA10
MCU_TX	PA9
SPI1_MOSI	PA7
SPI1_CLI	PA5

Note the following requirements:
- o All nets are required to have net names.
- o Add Non-Specific No ERC connectors ☒ on each pin that is unused.

b. Complete 02_Power.SchDoc

- Add a single instance of the TLV73333, along with any supporting passive components, required to generate a 3.3V output from the 5.0V power rail. You can use the "Typical Application Circuit" from page 1 of datasheet to determine how to properly connect the device.

Note the following requirements:

- o Set Cin and Cout to be 10uF capacitors.
- o Connect the Enable pin to 5.0V
- o Add Test Points for 5.0V and 3.3V
- o All nets are required to have net names.
- o Add Non-Specific No ERC connectors ☒ on each pin that is unused.

c. Annotate your schematics

In order for you to transfer our schematic design to a PCB document, every component must have a unique reference designator. Altium provides tools to do this for us.

Watch the following video that will instruct you how to annotate your schematics.

[Annotating Schematics](#)

1. Select **Tools→Annotate Schematics**
2. Altium will open a window that will show you how it will auto number your reference designators.
1. Select **Update Change List**, then OK
2. Select **Accept Changes**
3. In the pop up window, **Select Validate Changes**, then **Execute Changes**, then **Close**. If Altium detected any errors that need to be fixed, the green check mark would be a red X indicating you have to fix something.

4. Post Lab: What to Turn in

File Name	Description
LastName_FirstName_Lab2_Altium.zip	Altium Project ZIP file.
LastName_FirstName_Lab2_Sch.pdf	PDF of **completed** schematics Use File→Smart PDF to generate the schematics. You can use the default settings, but only include the .SchDocs on the 3rd page of the dialog.