

	1	2	3	4	5		
A		<div>Power Input</div> <div>Power_Input.sch</div> <div>POS3P3_Power Supply</div> <div>POS3P3_Power_Supply.sch</div> <div>Microcontroller Programming</div> <div>Microcontroller_Programming.sch</div> <div>WiFi Module</div> <div>WI_FI_Module.sch</div> <div>USB_UART Isolation</div> <div>USB_UART_Isolation.sch</div> <div>USB_UART Bridge</div> <div>USB_UART_Bridge.sch</div> <div>Panel Data Connectors</div> <div>Panel_Data_Connectors.sch</div> <div>LED_POS5 Monitoring</div> <div>LED_POS5_Monitoring.sch</div> <div>Microcontroller Power</div> <div>Microcontroller_Power.sch</div> <div>Microcontroller A</div> <div>Microcontroller_A.sch</div> <div>Microcontroller B</div> <div>Microcontroller_B.sch</div>		<div>External SRAM</div> <div>External_SRAM.sch</div> <div>External Flash 1</div> <div>External_Flash_1.sch</div> <div>External Flash 2</div> <div>External_Flash_2.sch</div> <div>External Flash 3</div> <div>External_Flash_3.sch</div> <div>External Flash 4</div> <div>External_Flash_4.sch</div> <div>External Flash 5</div> <div>External_Flash_5.sch</div> <div>External Flash 6</div> <div>External_Flash_6.sch</div> <div>External Flash 7</div> <div>External_Flash_7.sch</div> <div>External Flash 8</div> <div>External_Flash_8.sch</div> <div>Status_LEDs_1</div> <div>Status_LEDs_1.sch</div> <div>Panel_Data_Level Shifters 1</div> <div>Panel_Data_Level_Shifters_1.sch</div> <div>Panel_Data_Level Shifters 2</div> <div>Panel_Data_Level_Shifters_2.sch</div> <div>Panel_Data_Level Shifters 3</div> <div>Panel_Data_Level_Shifters_3.sch</div> <div>Test Points</div> <div>Test_Points.sch</div> <div>Pushbuttons</div> <div>Pushbuttons.sch</div> <div>POS5_Power Supply</div> <div>POS5_Power_Supply.sch</div>	<div>Mechanical</div> <div>Mechanical.sch</div> <div>Internal_Rail Monitoring</div> <div>Internal_Rail_Monitoring.sch</div> <div>Status_LEDs_2</div> <div>Status_LEDs_2.sch</div>		A
B						B	
C						C	
D						D	

To Do List:

* Add modifications to power board KiCAD Project

* Add MU Logo to each sheet

* Add Titles to each sheet

* Re-order sheets

* Assign Refdes's

* Verify pinouts

* Verify peripheral wiring on micro

* Draw custom footprints

* Assign footprints

* Assign Digi-Key Partnumbers

* Run ERC, resolve errors

* Generate netlist

* Generate BOM

* Layout PCB

Sheet: /

File: LED_Display_Controller.sch

Title:

Size: A

Date:

KiCad E.D.A. kicad (5.0.1)–3

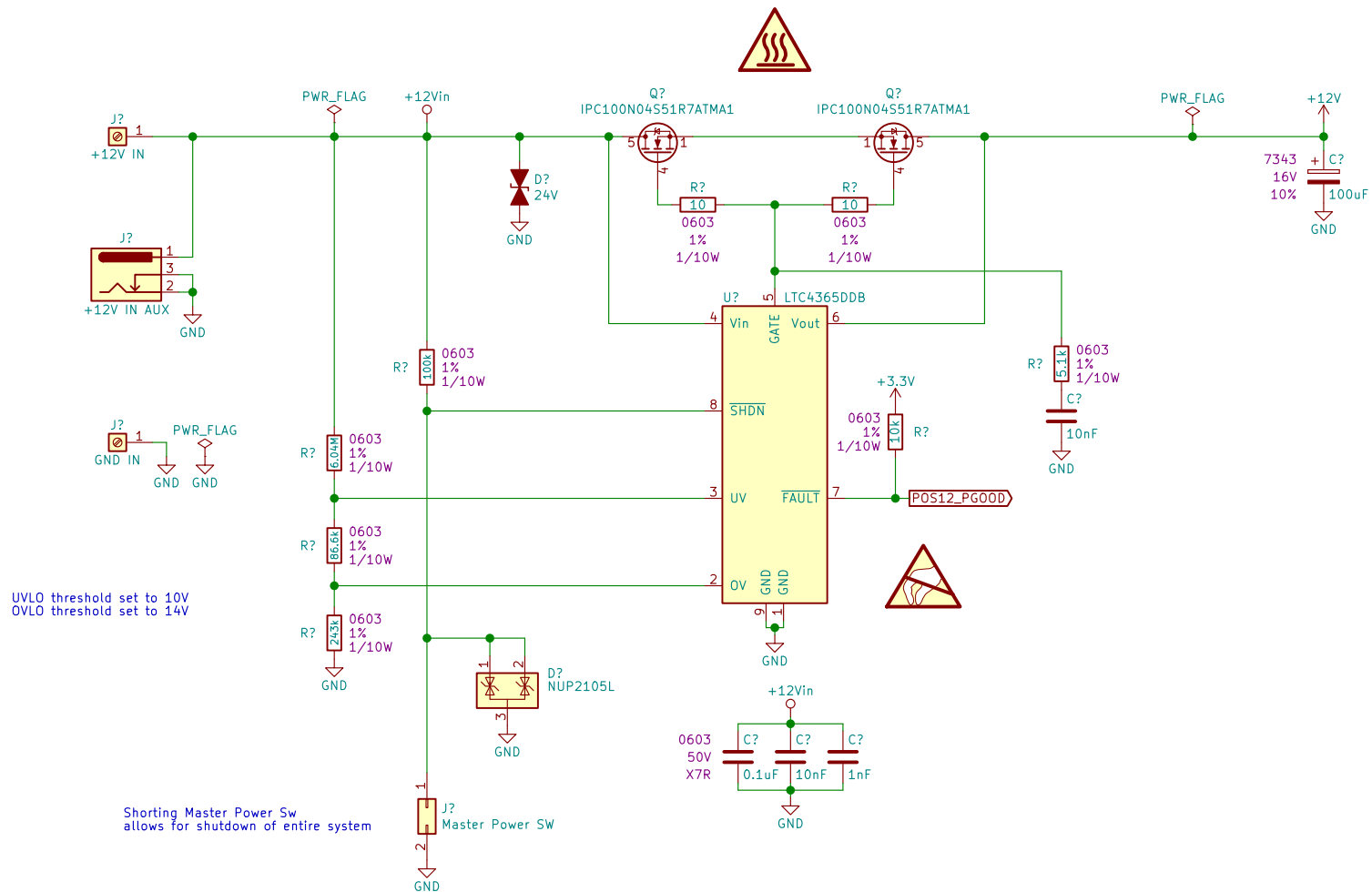
Rev:

Id: 1/31

- To Do List:
- * Add modifications to power board KICAD Project
 - * Add MU Logo to each sheet
 - * Add Titles to each sheet
 - * Re-order sheets
 - * Assign Refdes's
 - * Verify pinouts
 - * Verify peripheral wiring on micro
 - * Draw custom footprints
 - * Assign footprints
 - * Assign Digi-Key Partnumbers
 - * Run ERC, resolve errors
 - * Generate netlist
 - * Generate BOM
 - * Layout PCB

Sheet: /		
File: LED_Display_Controller.sch		
Title:		
Size: A	Date:	Rev:
KiCad E.D.A. kicad (5.0.1)-3		Id: 1/31

Power Input



Sheet: /Power Input/
File: Power_Input.sch

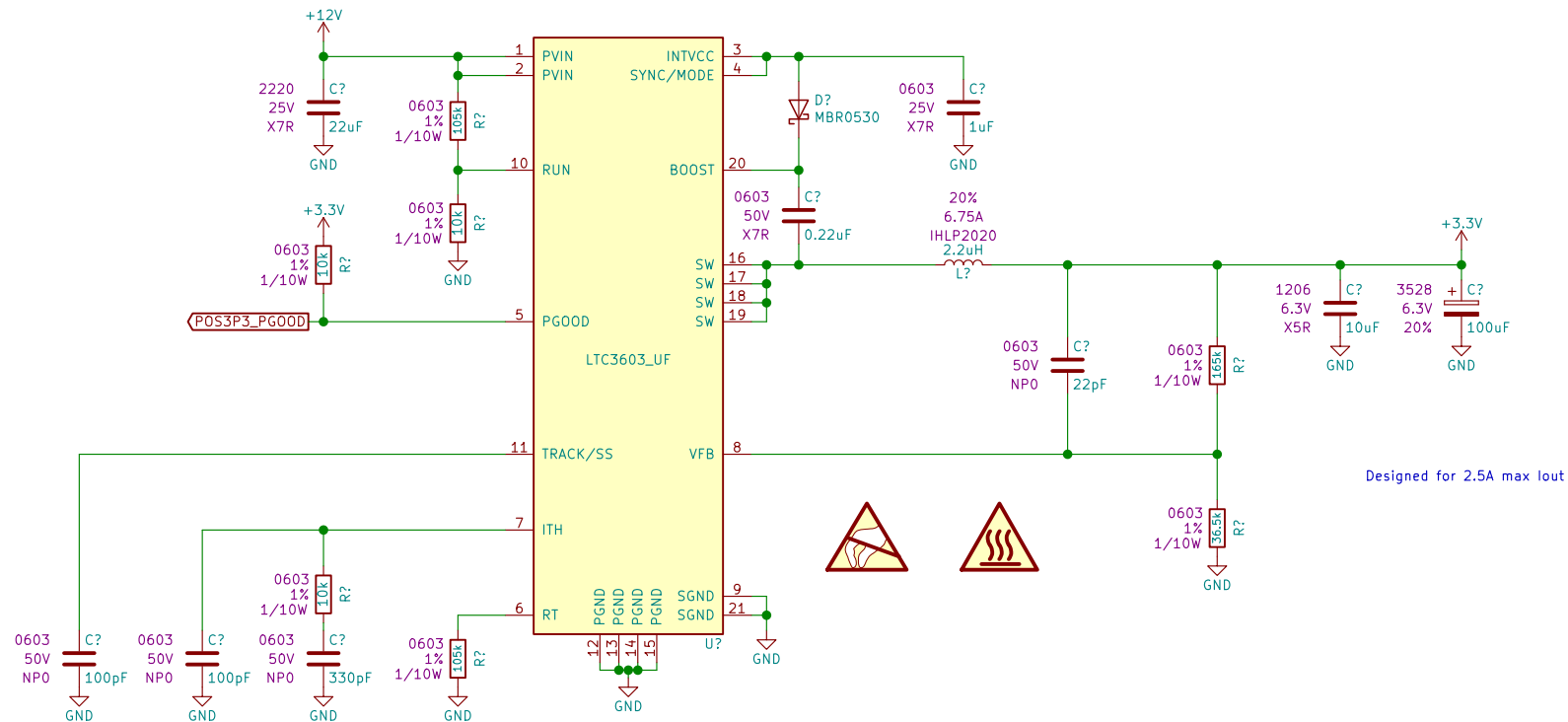
Title:

Size: A	Date:
KiCad E.D.A. kicad (5.0.1)-3	

Date:

Rev:

Id: 2/31



Designed for 2.5A max iout

Sheet: /POS3P3 Power Supply/
File: POS3P3_Power_Supply.sch

Title:

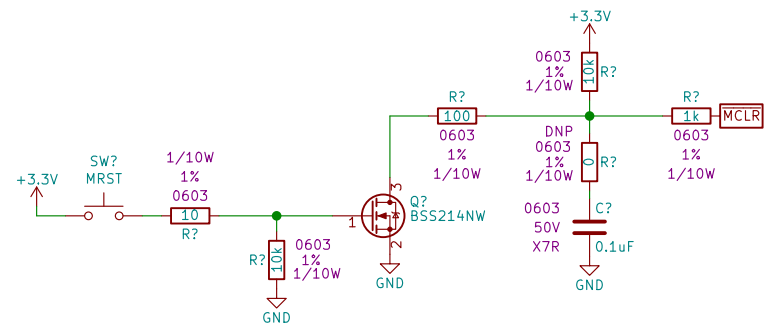
Size: A

Date:

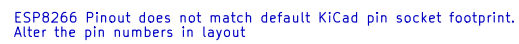
KiCad E.D.A. kicad (5.0.1)–3

Rev:

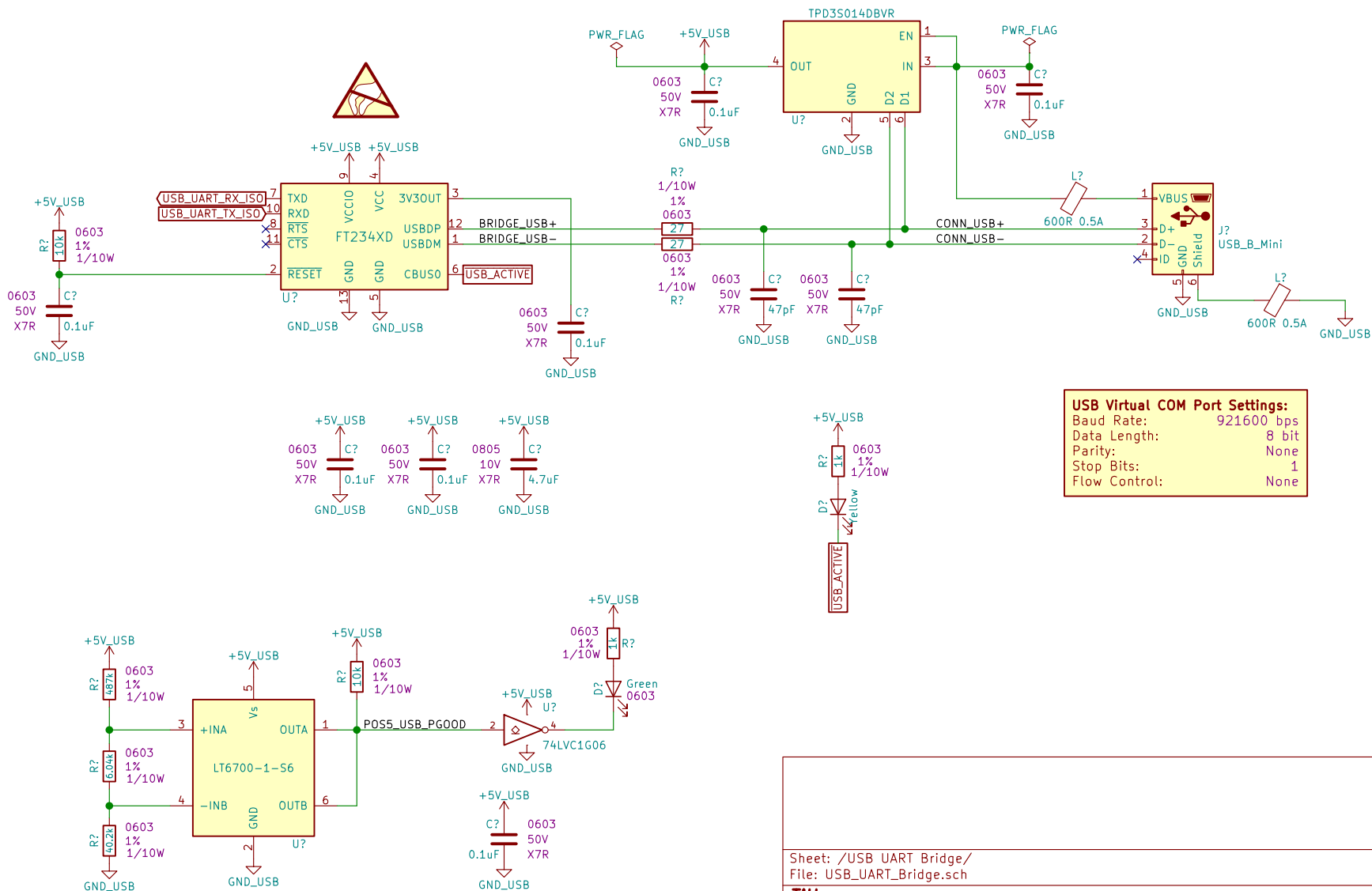
Id: 3/31



Id: 4/31



Id: 5/31

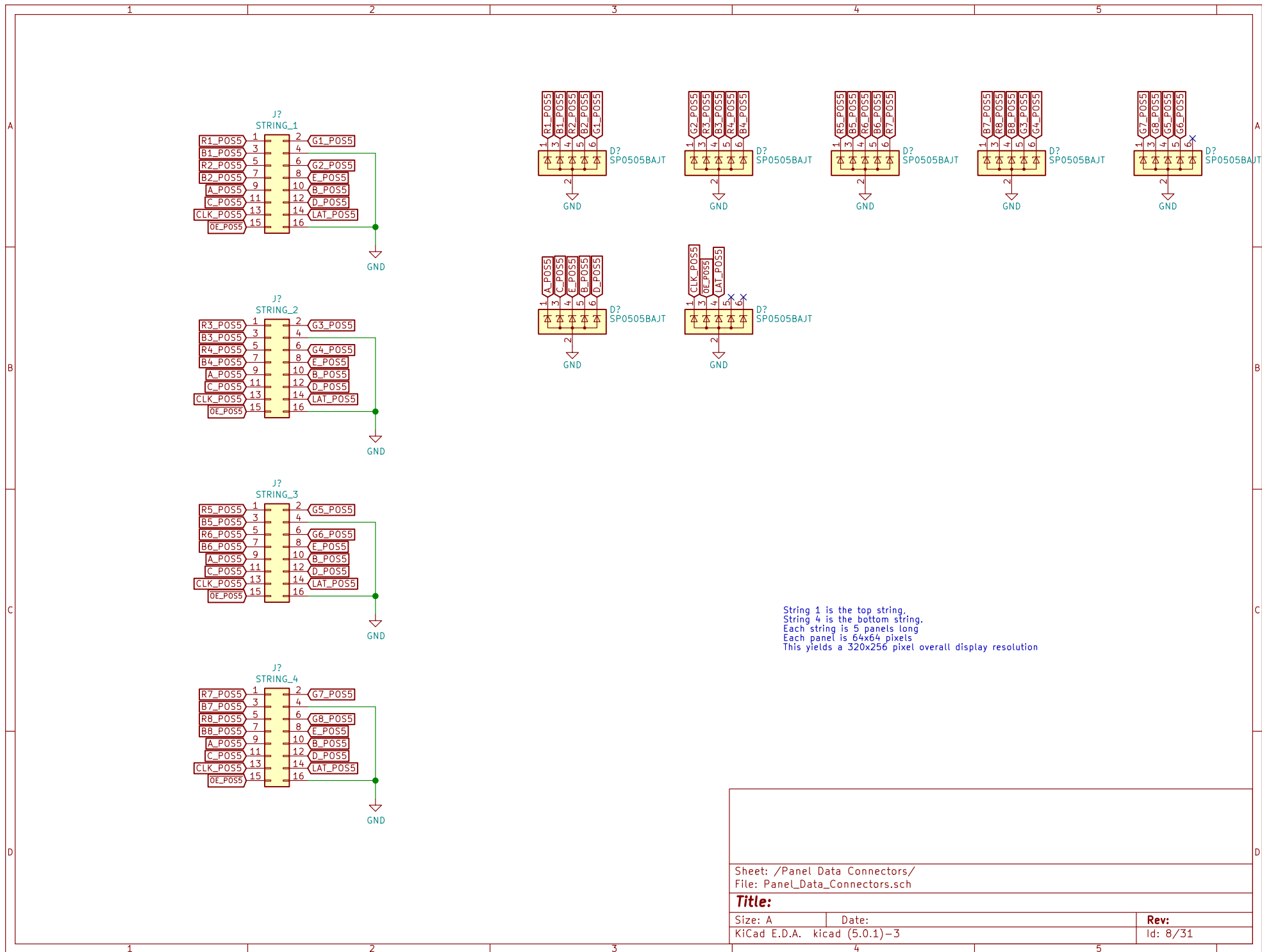


Sheet: /USB UART Bridge/
File: USB_UART_Bridge.sch

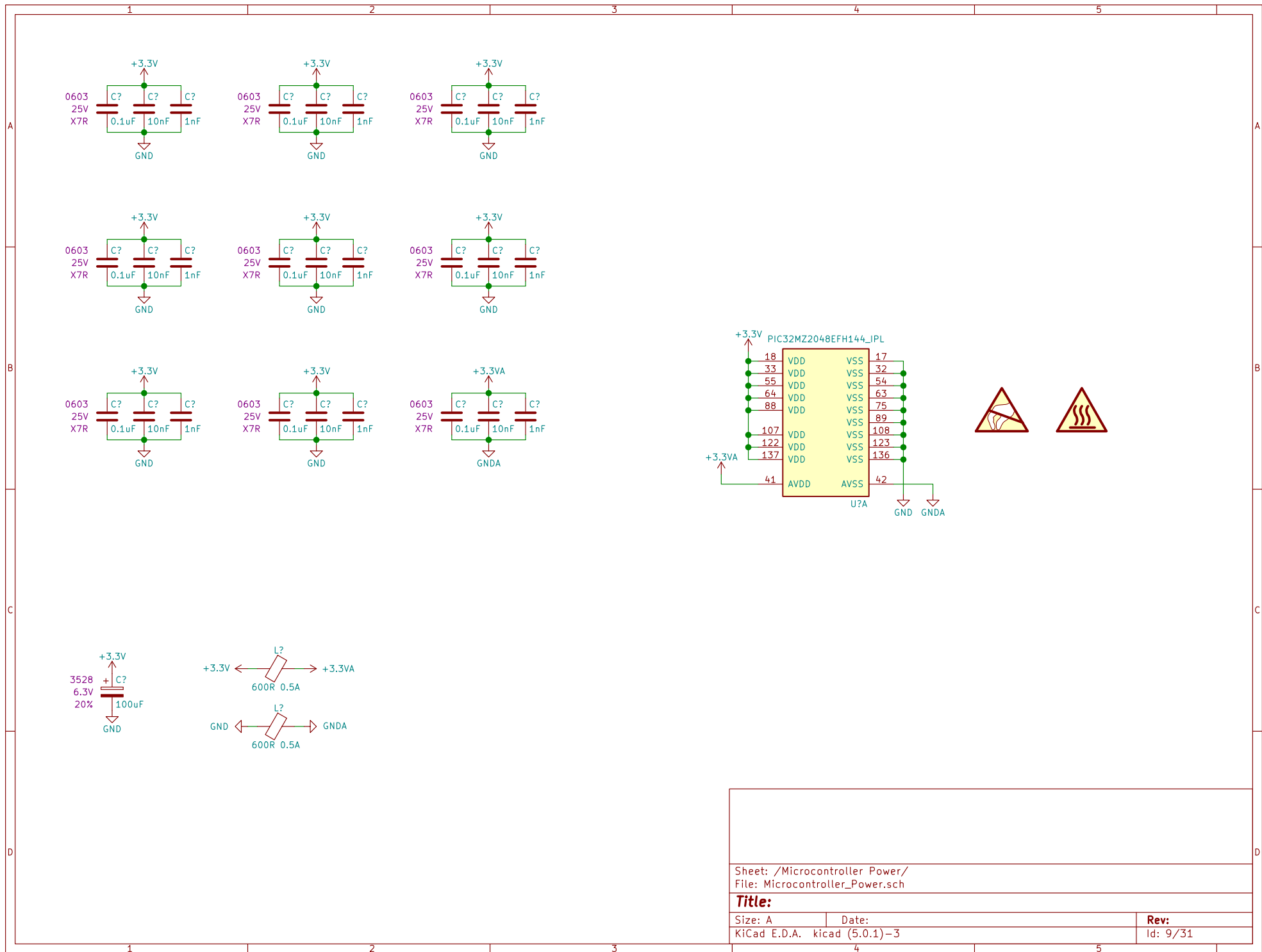
Title:

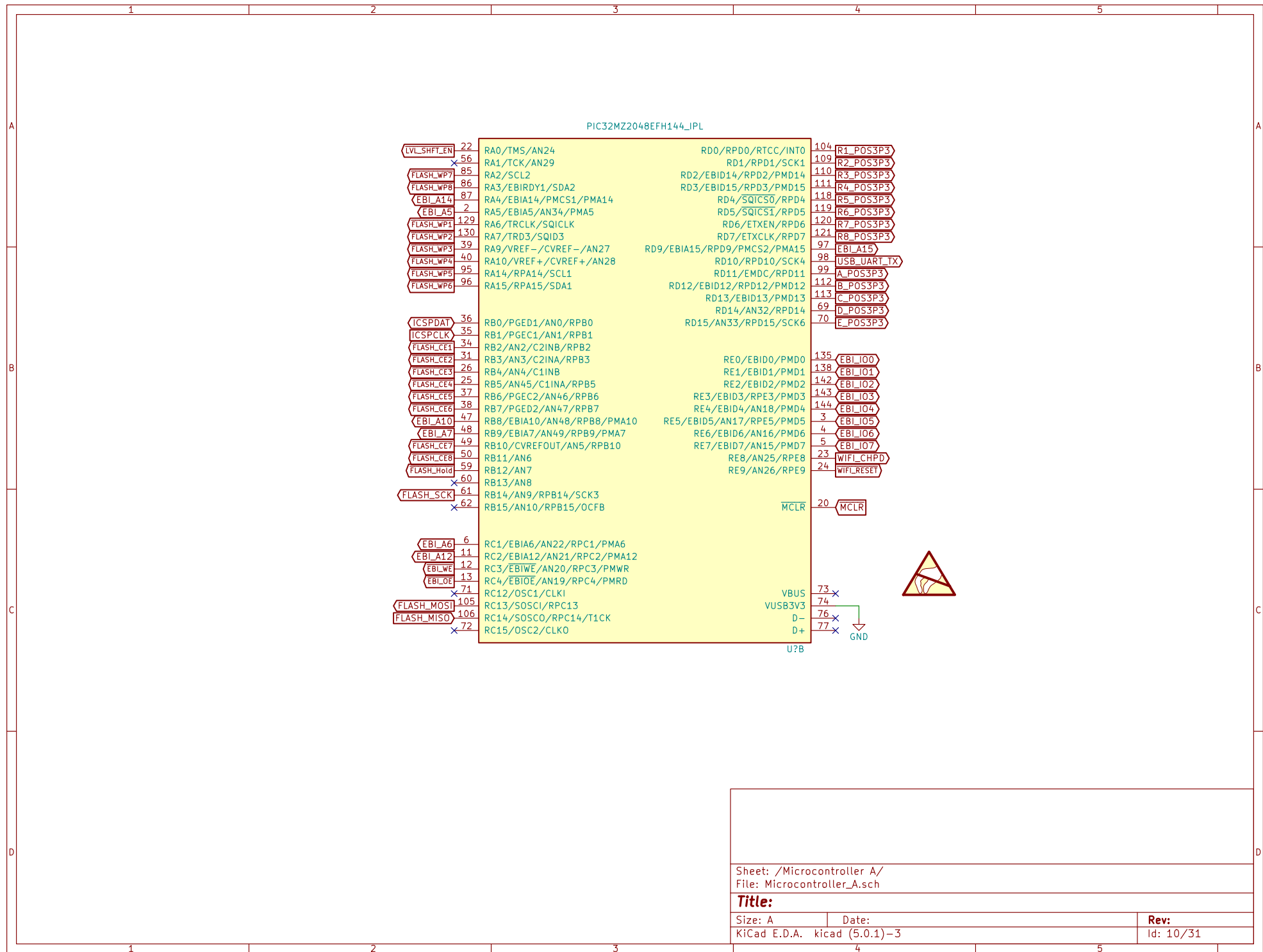
Size: A Date:
KiCad E.D.A. kicad (5.0.1)-3

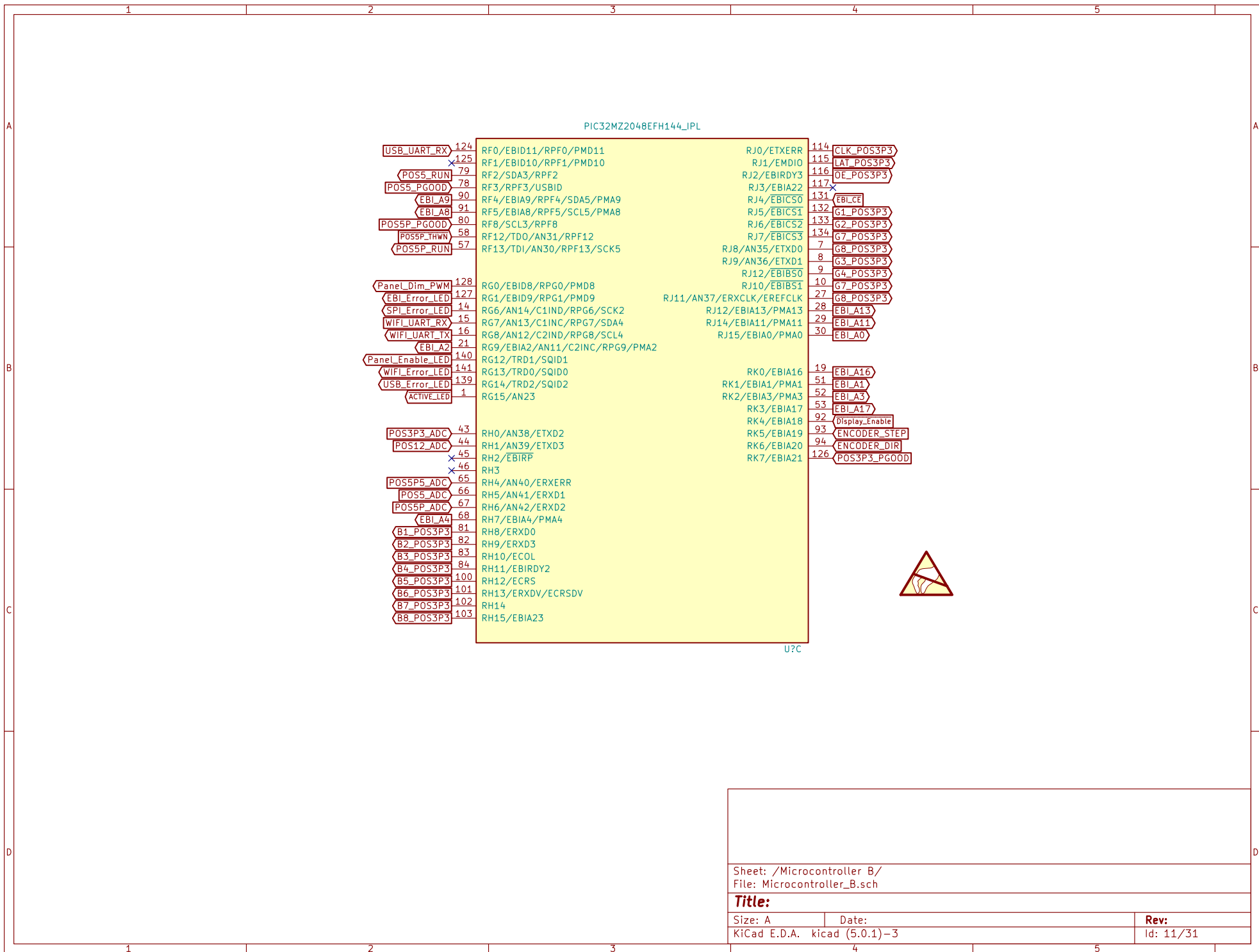
Rev:
Id: 7/31



Sheet: /Panel Data Connectors/ File: PanelData_Connectors.sch		
Title:		
Size: A	Date:	Rev:
KiCad E.D.A. kicad (5.0.1)–3		Id: 8/31







Sheet: /Microcontroller B/
File: Microcontroller_B.sch

Title:

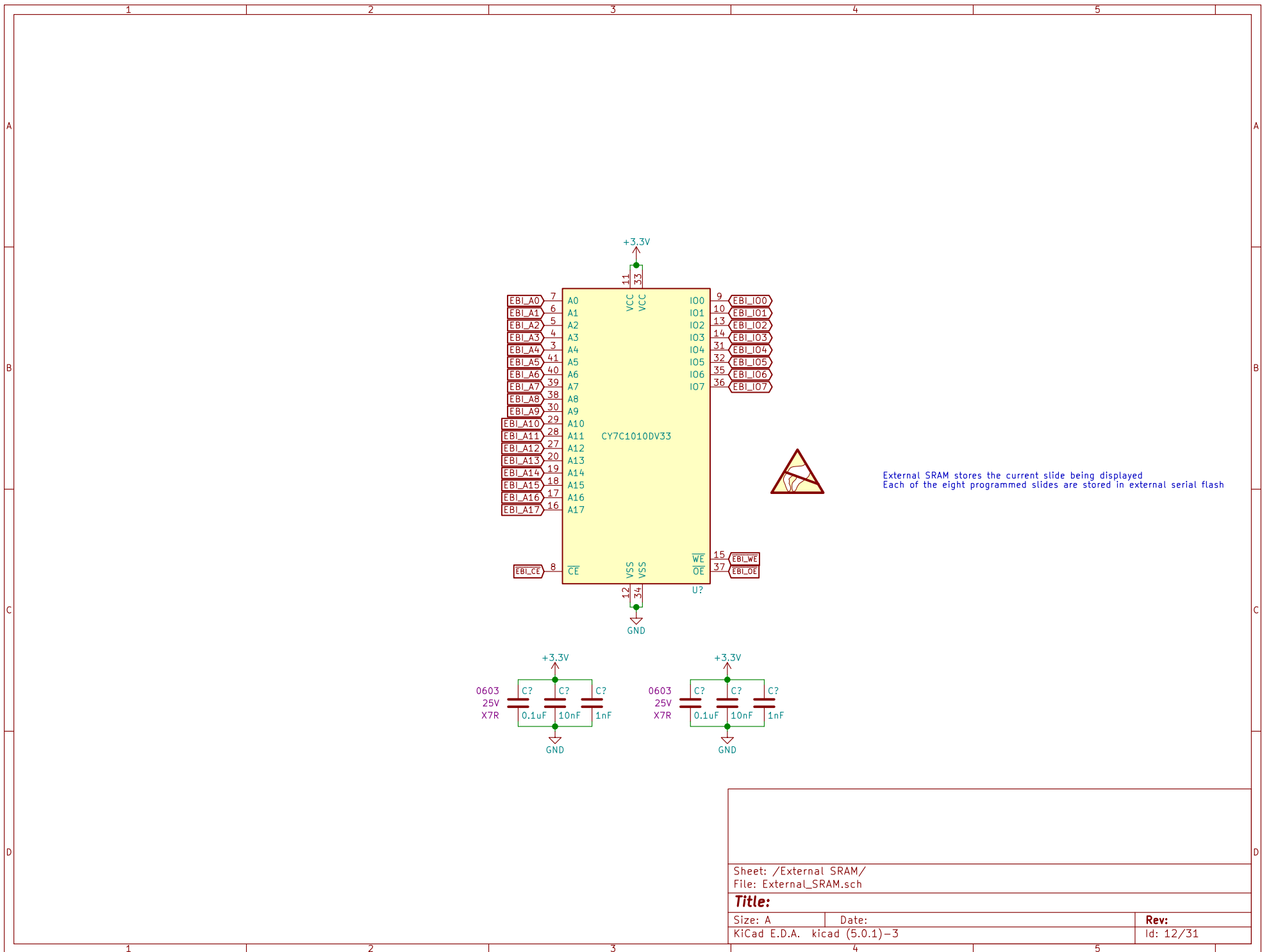
Size: A

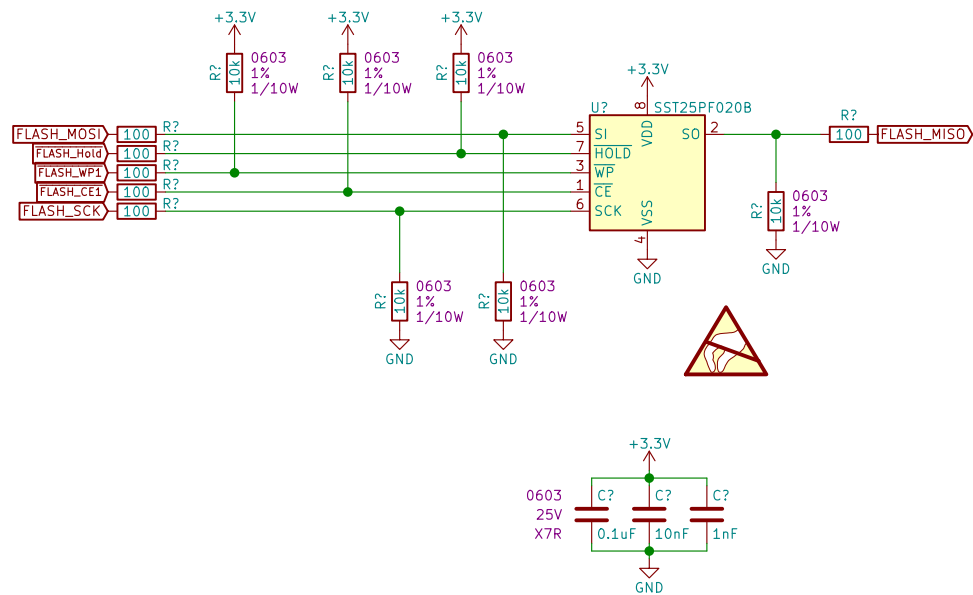
Date:

KiCad E.D.A. kicad (5.0.1)-3

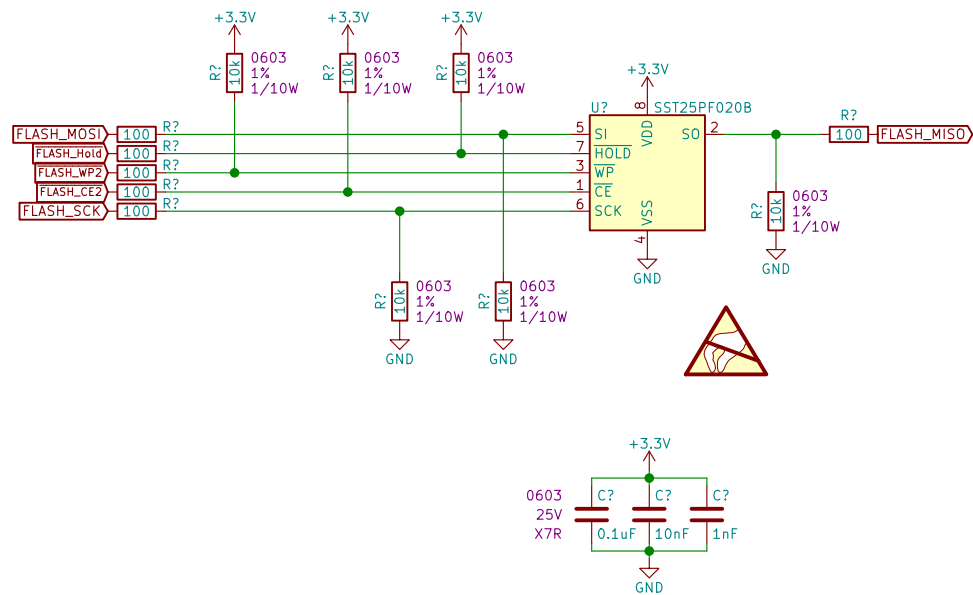
Rev:

Id: 11/31

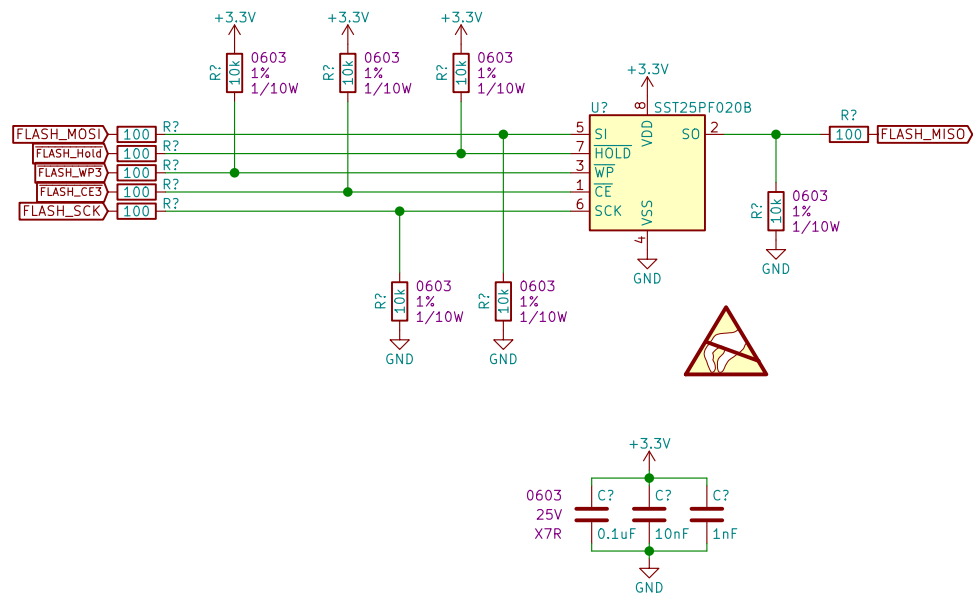




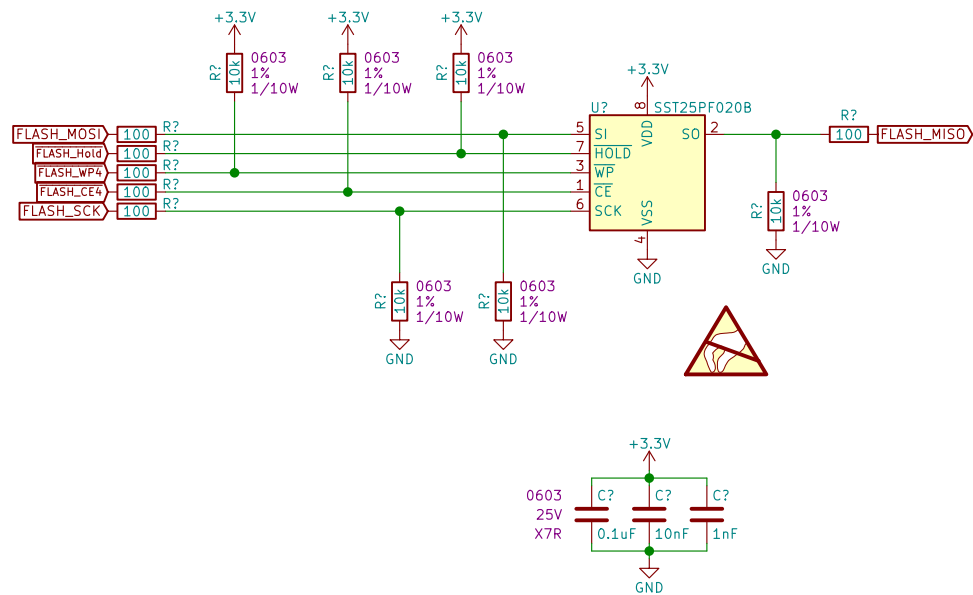
Sheet: /External Flash 1/ File: External_Flash_1.sch		
Title:		
Size: A	Date:	Rev:
KiCad E.D.A. kicad (5.0.1)-3		Id: 13/31



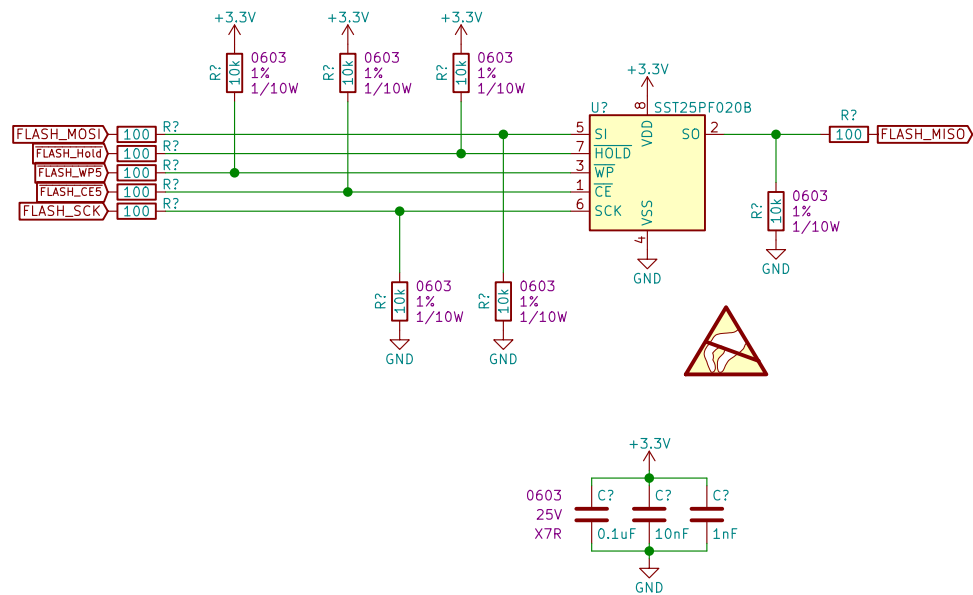
Sheet: /External Flash 2/ File: External_Flash_2.sch		
Title:		
Size: A	Date:	Rev:
KiCad E.D.A. kicad (5.0.1)-3		Id: 14/31



Sheet: /External Flash 3/ File: External_Flash_3.sch		
Title:		
Size: A	Date:	Rev:
KiCad E.D.A. kicad (5.0.1)-3		Id: 15/31



Sheet: /External Flash 4/ File: External_Flash_4.sch		
Title:		
Size: A	Date:	Rev:
KiCad E.D.A. kicad (5.0.1)-3		Id: 16/31



Sheet: /External Flash 5/
File: External_Flash_5.sch

Title:

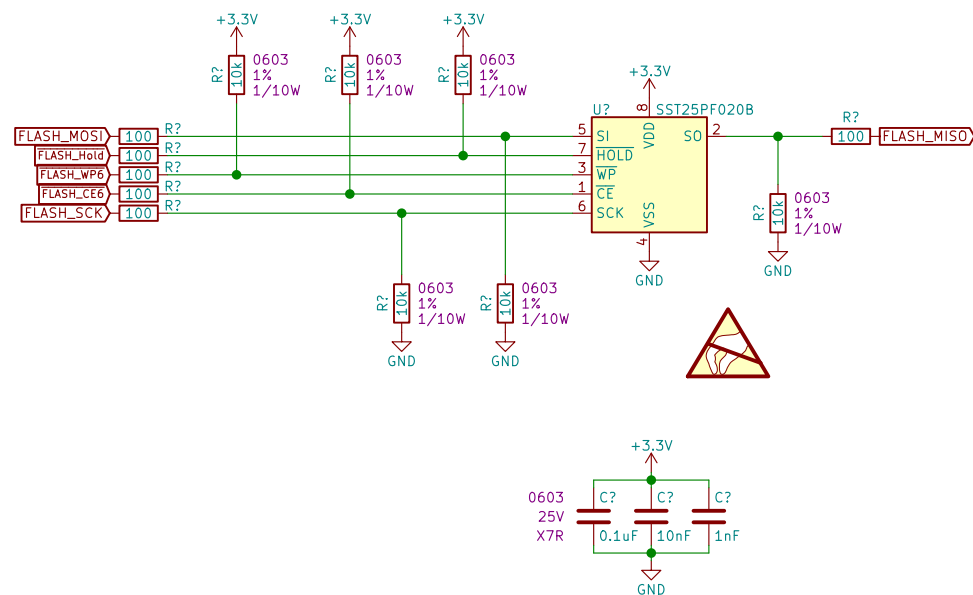
Size: A

Date:

KiCad E.D.A. kicad (5.0.1)-3

Rev:

Id: 17/31



Sheet: /External Flash 6/
File: External_Flash_6.sch

Title:

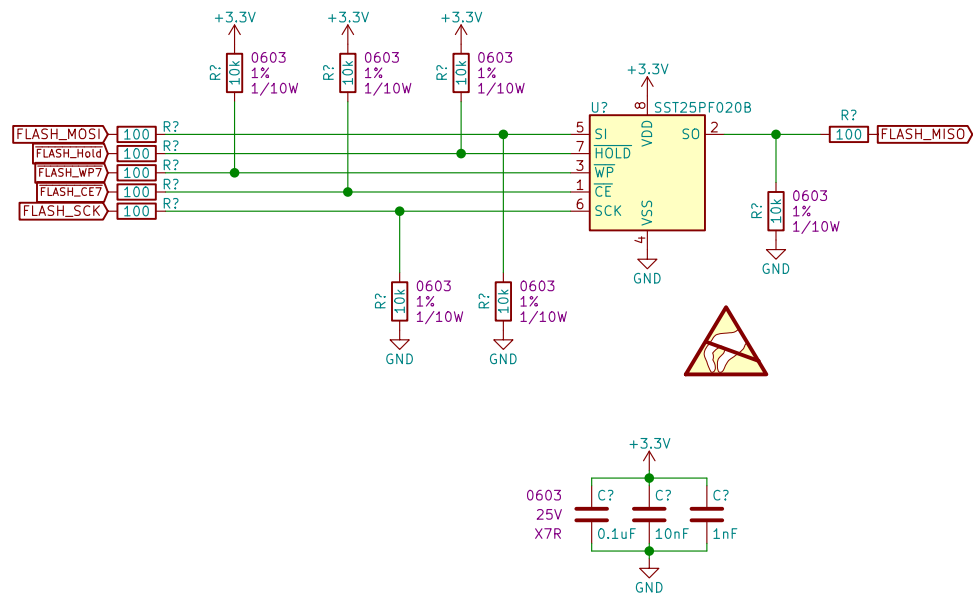
Size: A

Date:

KiCad E.D.A. kicad (5.0.1)-3

Rev:

Id: 18/31



Sheet: /External Flash 7/
File: External_Flash_7.sch

Title:

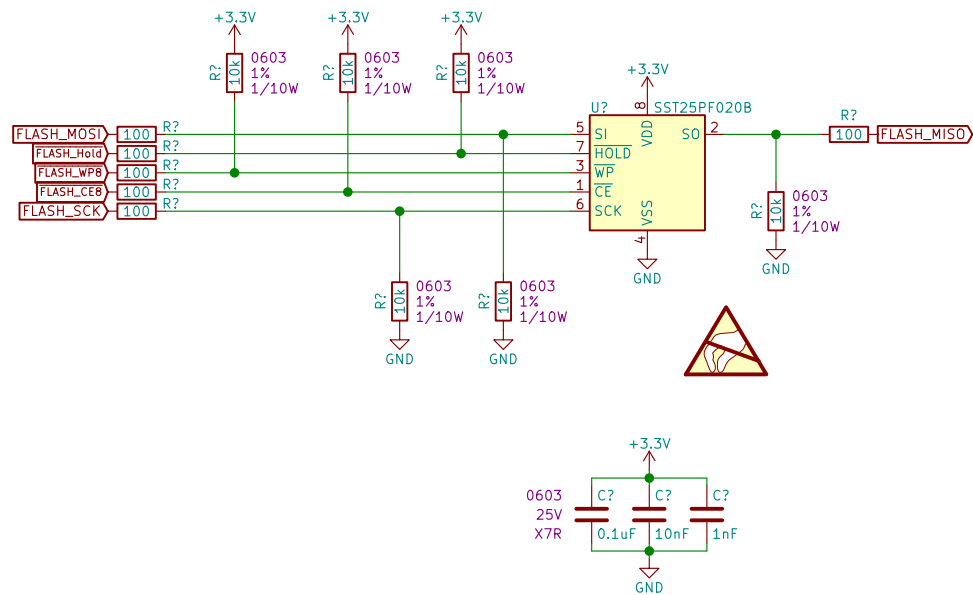
Size: A

Date:

KiCad E.D.A. kicad (5.0.1)-3

Rev:

Id: 19/31



Sheet: /External Flash 8/
File: External_Flash_8.sch

Title:

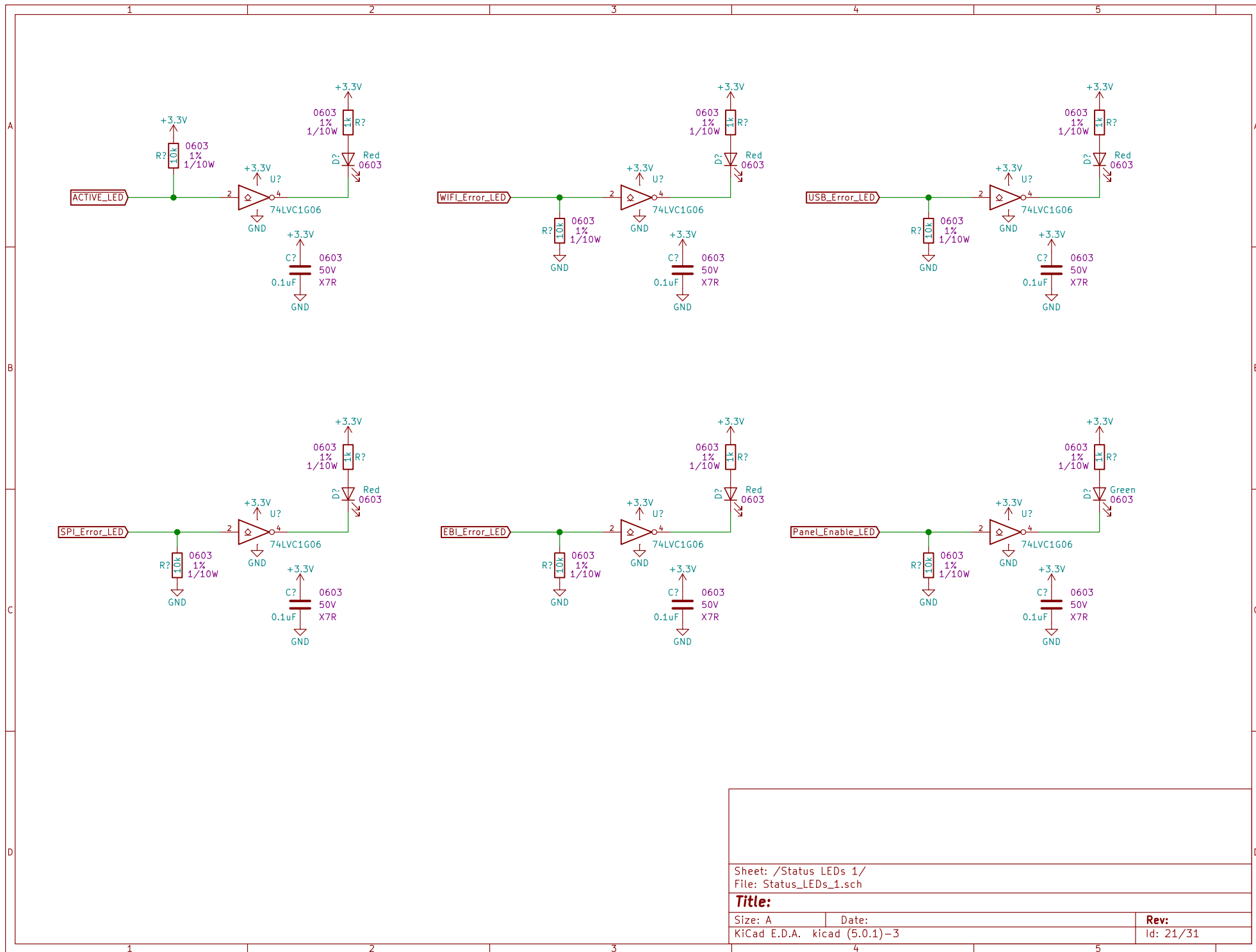
Size: A

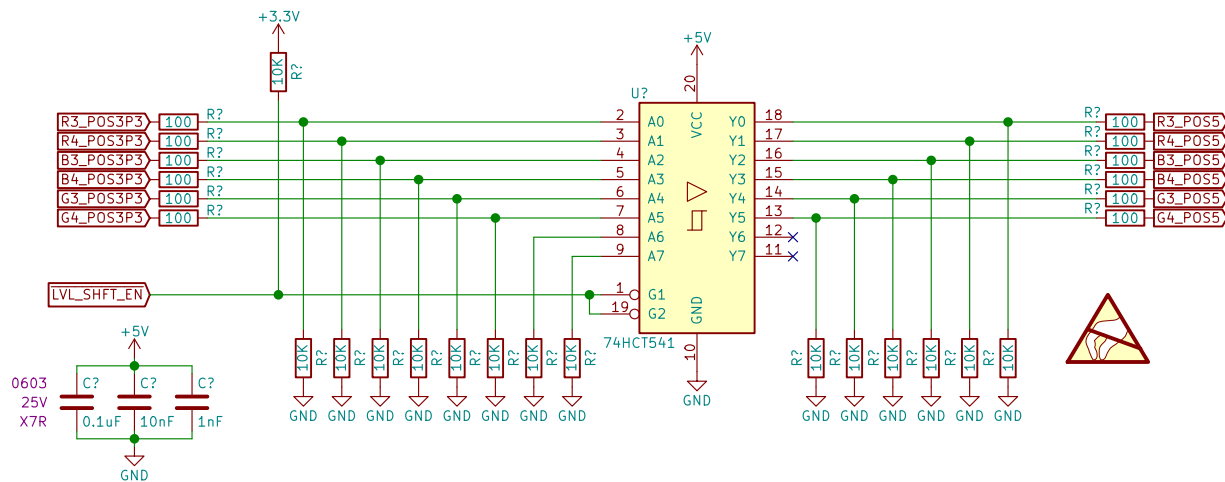
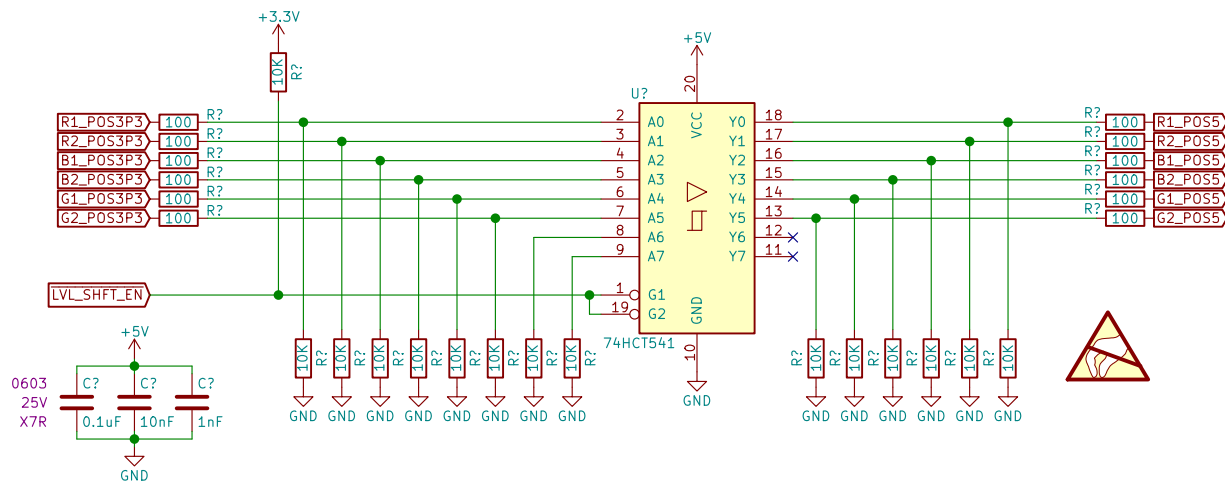
Date:

KiCad E.D.A. kicad (5.0.1)–3

Rev:

Id: 20/31





Sheet: /Panel Data Level Shifters 1/
File: PanelData_LevelShifters_1.sch

Title:

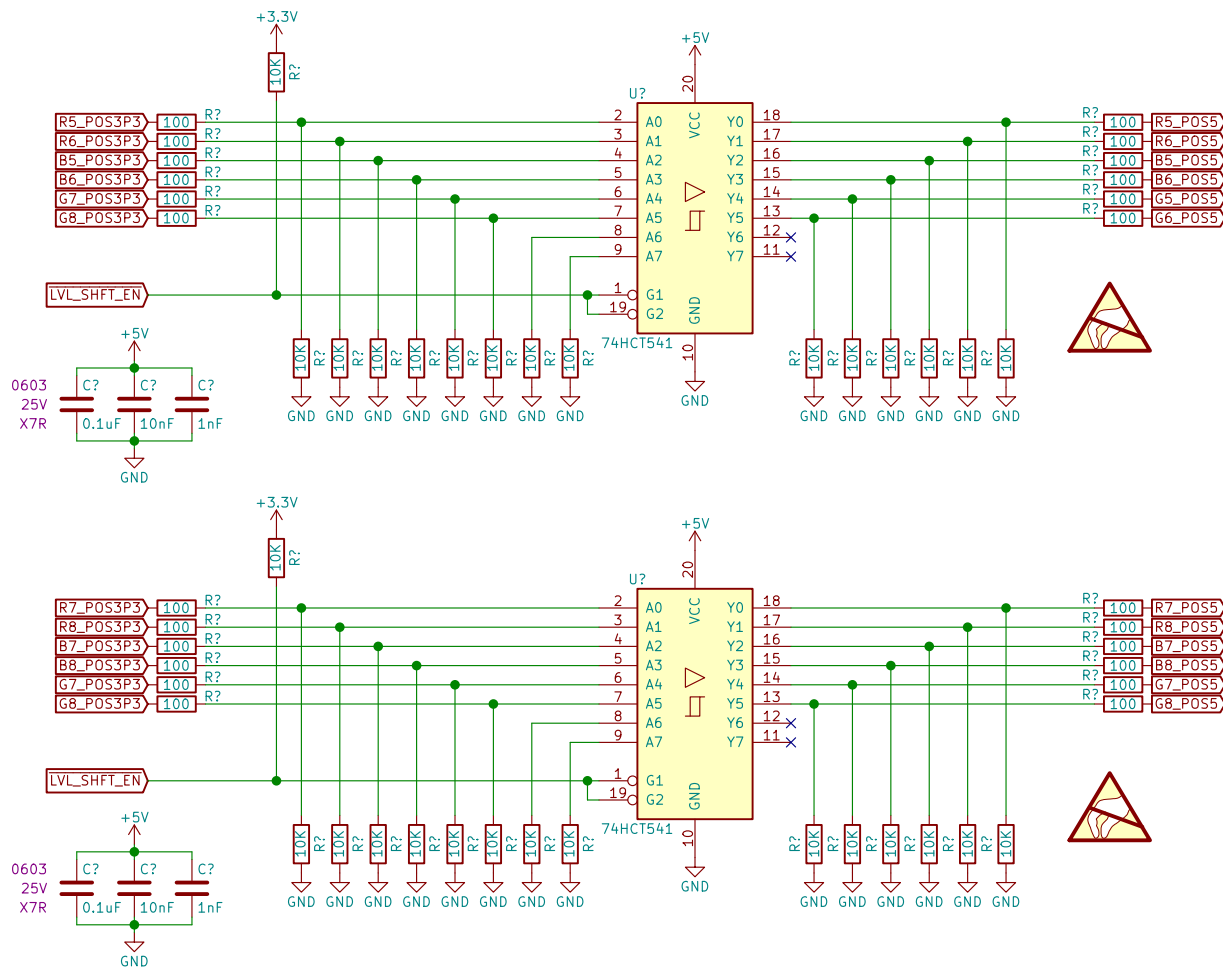
Size: A

Date:

KiCad E.D.A. kicad (5.0.1)-3

Rev:

Id: 22/31



Sheet: /Panel Data Level Shifters 2/
File: PanelData_LevelShifters_2.sch

Title:

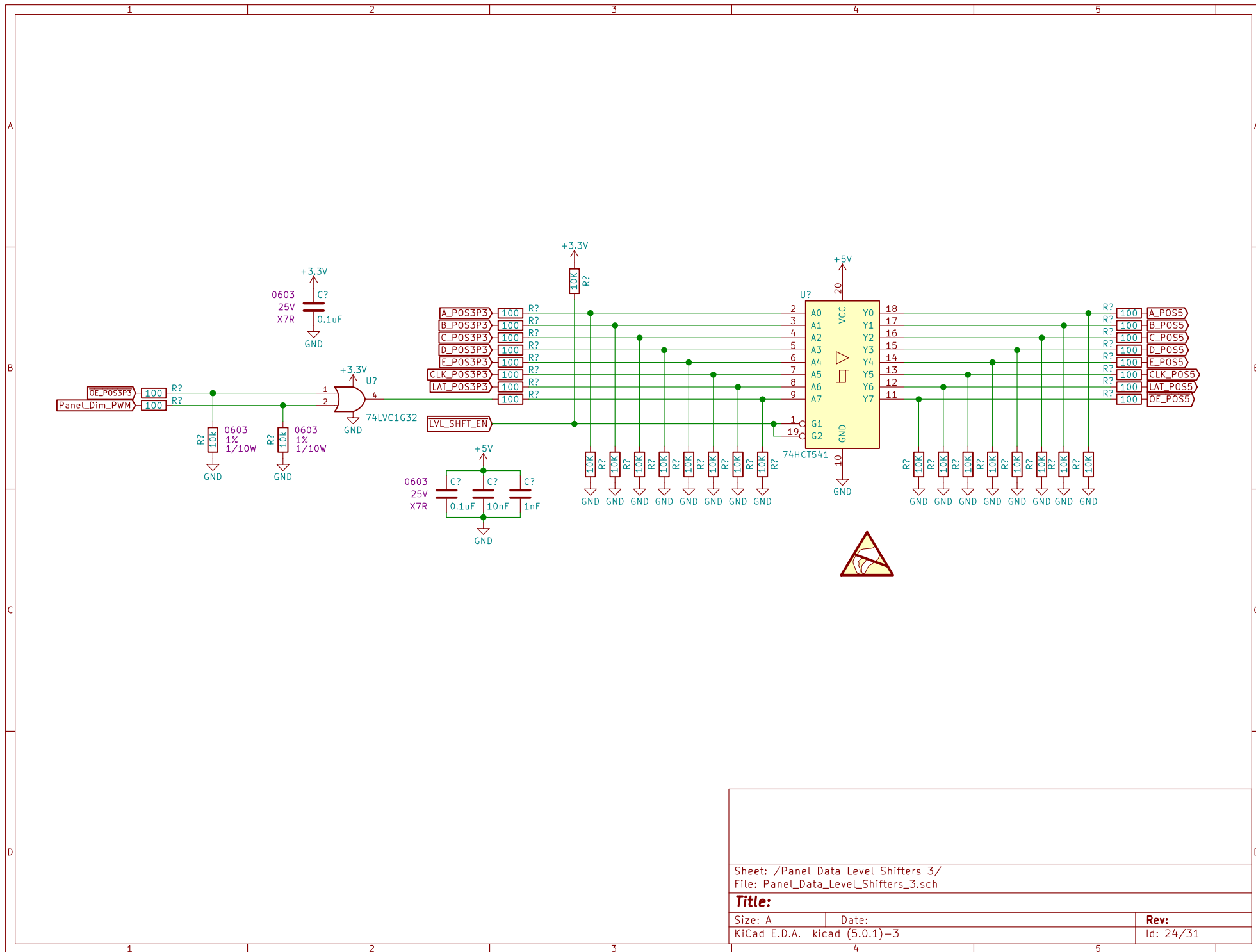
Size: A

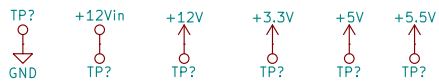
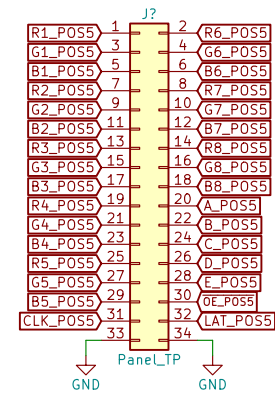
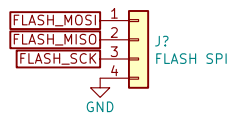
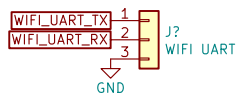
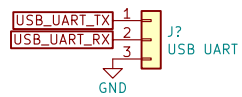
Date:

KiCad E.D.A. kicad (5.0.1)-3

Rev:

Id: 23/31





Sheet: /Test Points/
File: Test_Points.sch

Title:

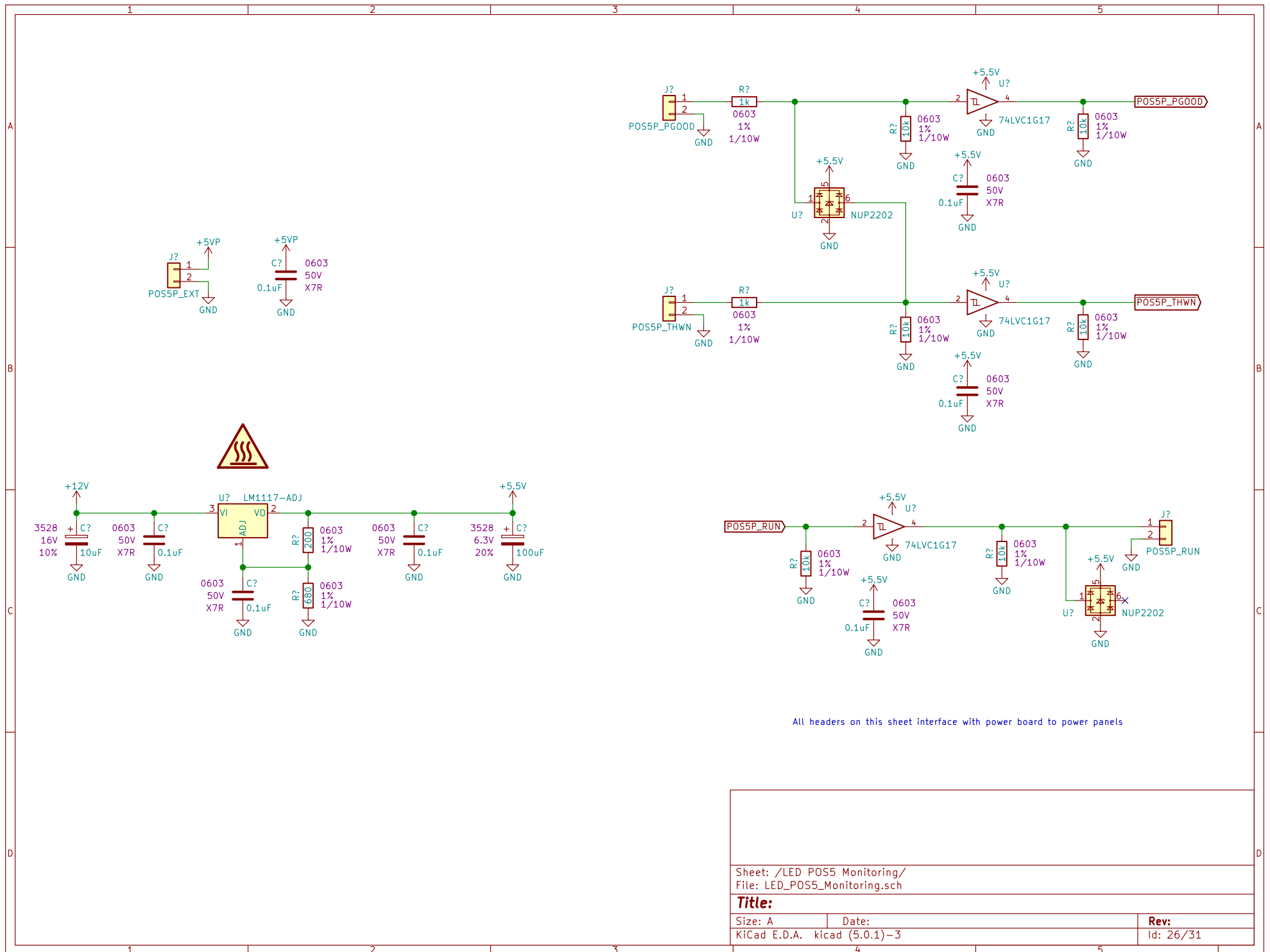
Size: A

Date:

KiCad E.D.A. kicad (5.0.1)-3

Rev:

Id: 25/31



Sheet: /LED POS5 Monitoring/
File: LED_POS5_Monitoring.sch

Title:

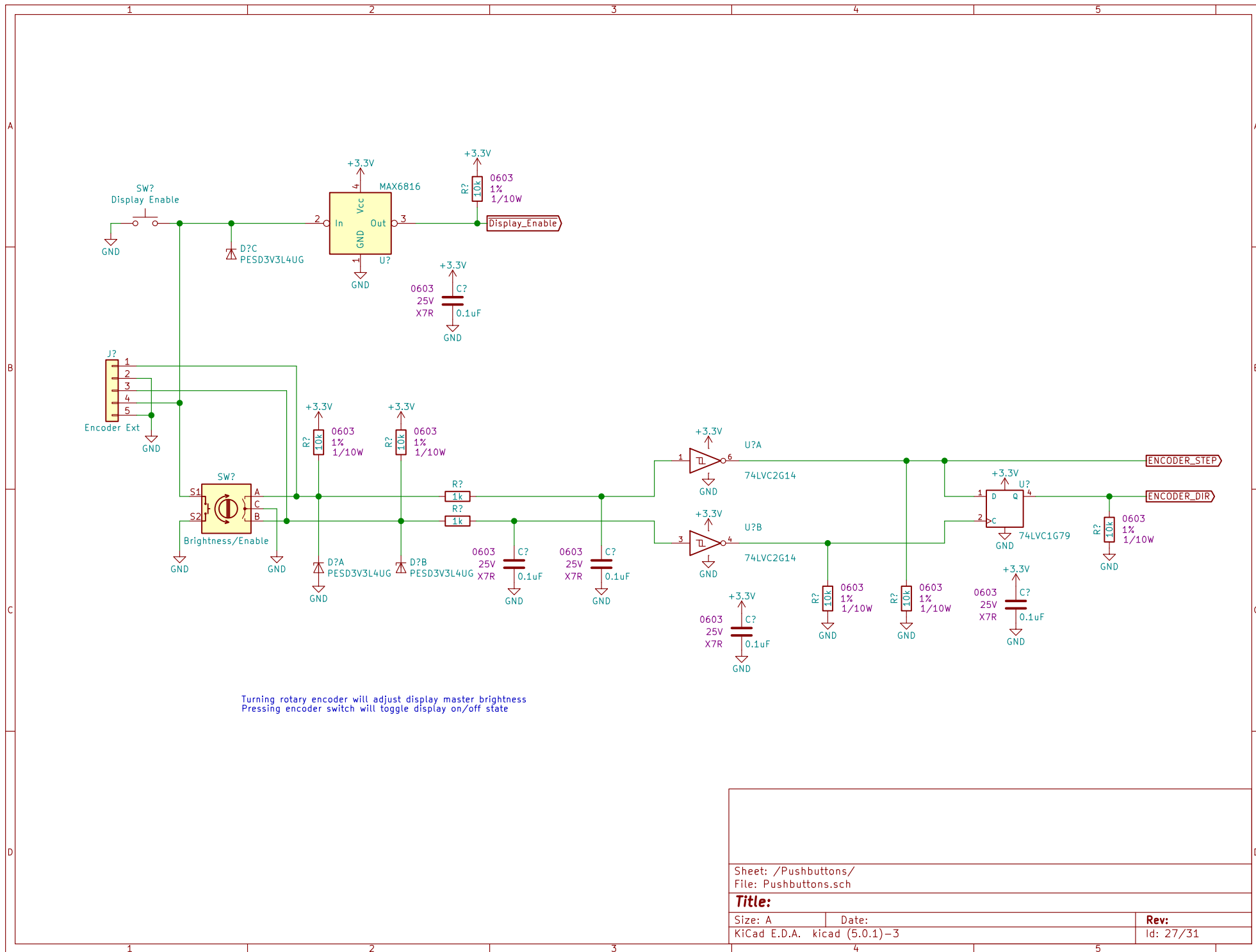
Size: A

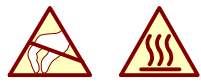
Date:

KiCad E.D.A. kicad (5.0.1)-3

Rev:

Id: 26/31





Sheet: /POS5 Power Supply/
File: POS5_Power_Supply.sch

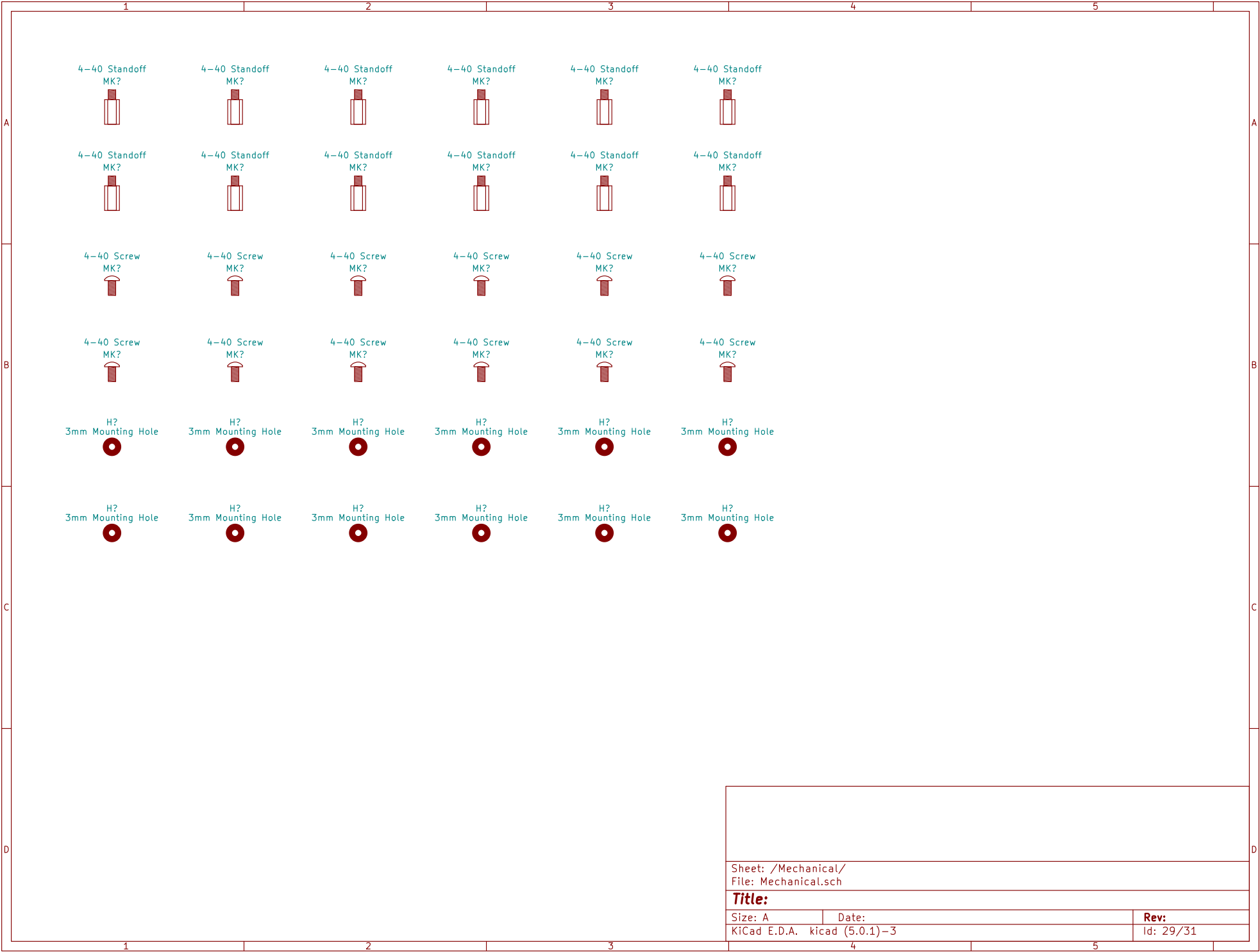
Size: A

Date:

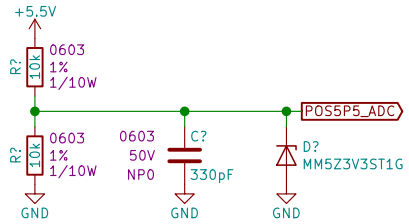
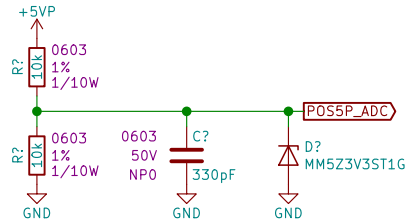
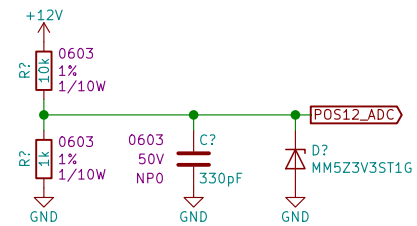
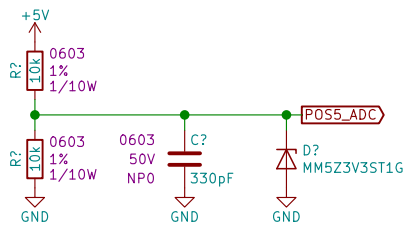
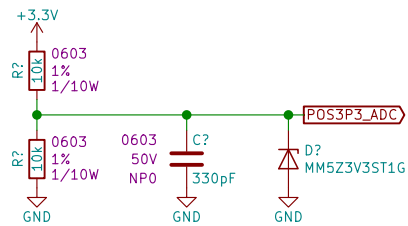
KiCad E.D.A.	kiCad (5.0.1)–3
--------------	-----------------

Rev:

Id: 28/31



Sheet: /Mechanical/ File: Mechanical.sch		
Title:		
Size: A	Date:	Rev:
KiCad E.D.A. kicad (5.0.1)-3		Id: 29/31



Sheet: /Internal Rail Monitoring/
File: Internal_Rail_Monitoring.sch

Title:

Size: A

Date:

KiCad E.D.A. kicad (5.0.1)-3

Rev:

Id: 30/31

