

	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
- * Add relevant design notes/routing notes to sheets
- * 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

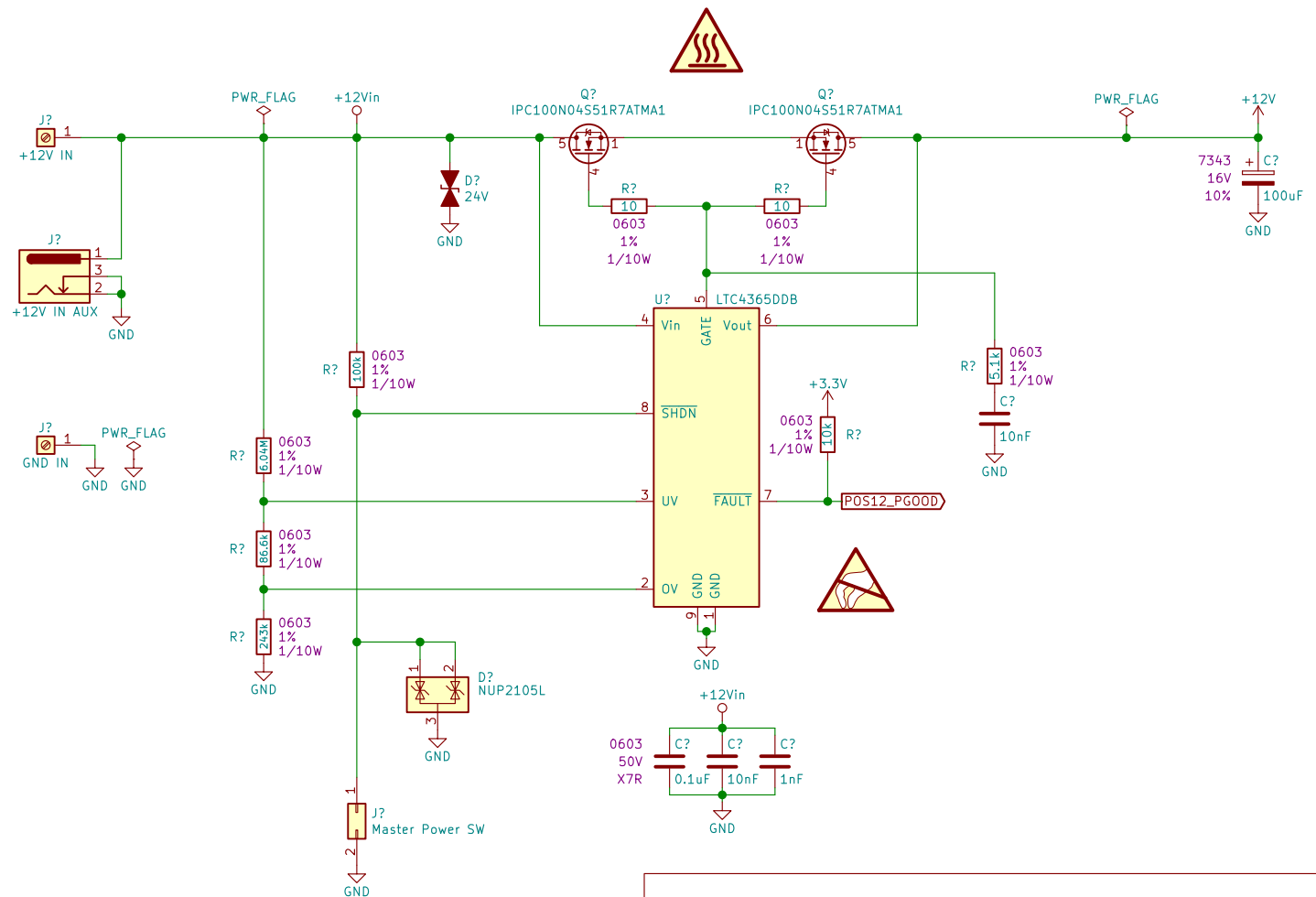
Rev:

Id: 1/31

- To Do List:
- * Add modifications to power board KiCAD Project
 - * Add MU Logo to each sheet
 - * Add Titles to each sheet
 - * Add relevant design notes/routing notes to sheets
 - * 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:

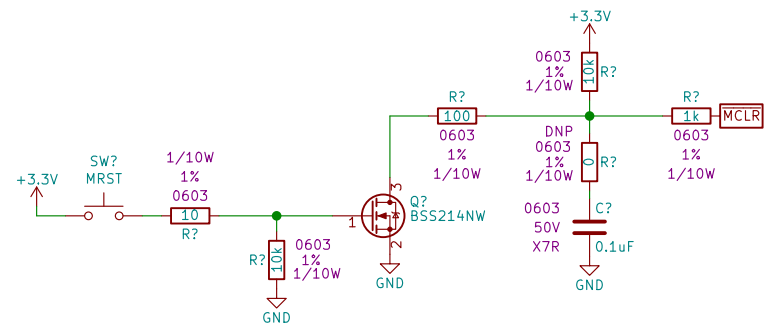
KiCad E.D.A. kicad (5.0.1)-3

Rev:

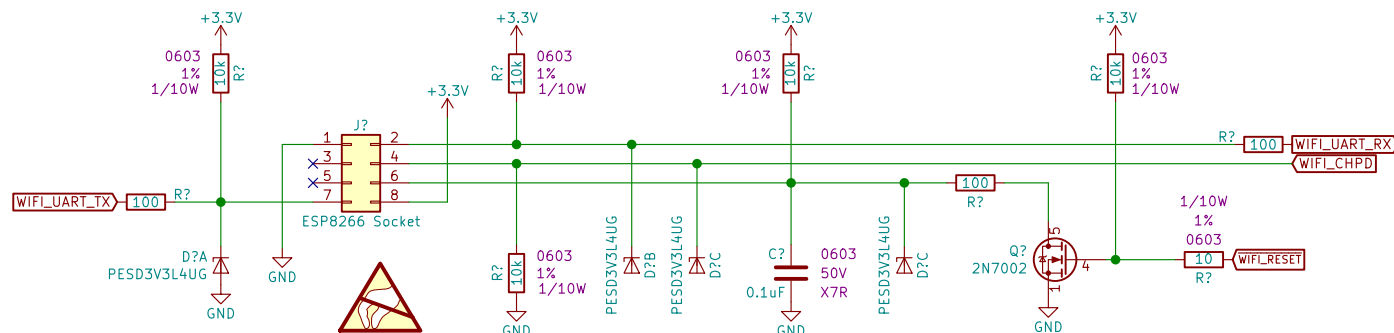
Id: 2/31



Id: 3/31



Size: A	Date:	Rev:
KiCad E.D.A. kicad (5.0.1)-3		Id: 4/31



ESP8266 Pinout does not match default KiCad pin socket footprint.
Alter the pin numbers in layout

Sheet: /WiFi Module/
File: Wi-Fi_Module.sch

Title:

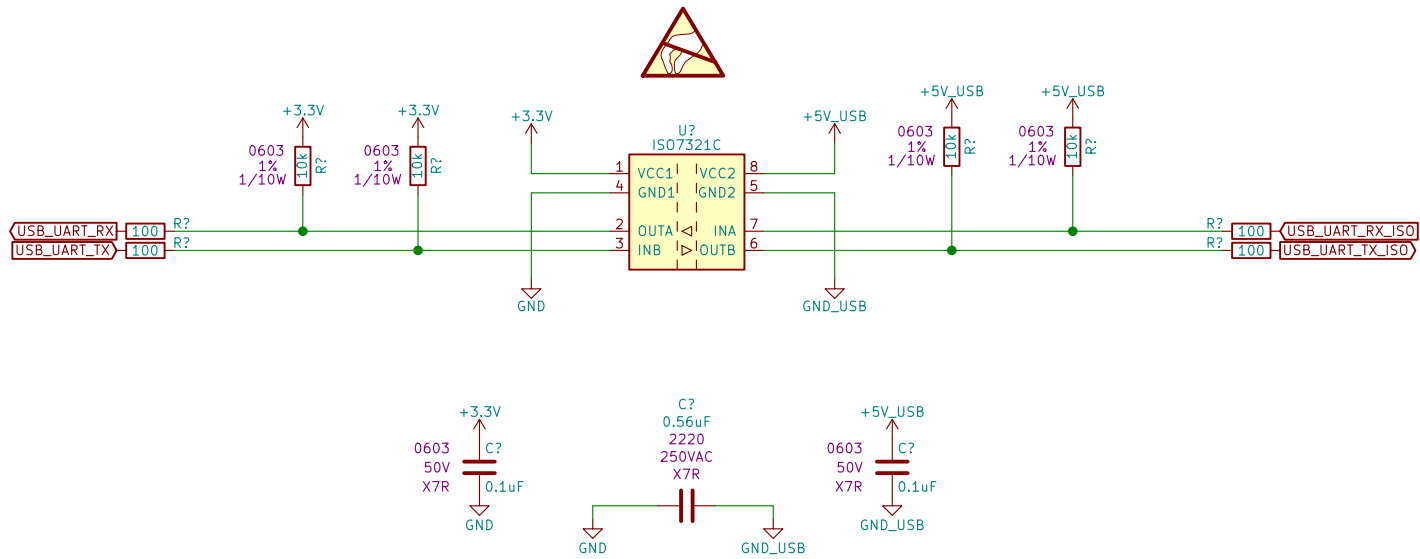
Size: A

Date:

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

Rev:

Id: 5/31



Sheet: /USB UART Isolation/
File: USB_UART_Isolation.sch

Title:

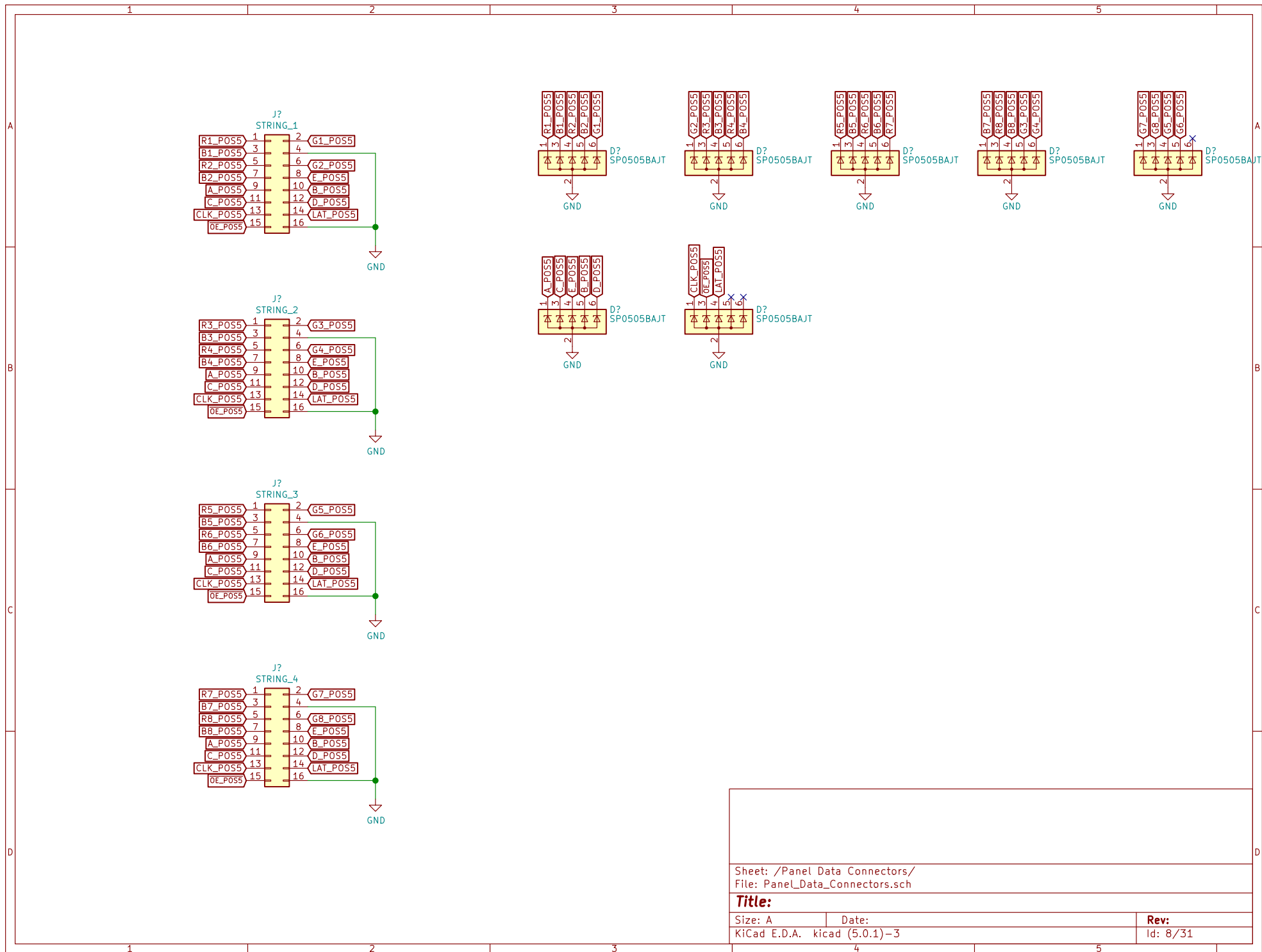
Size: A

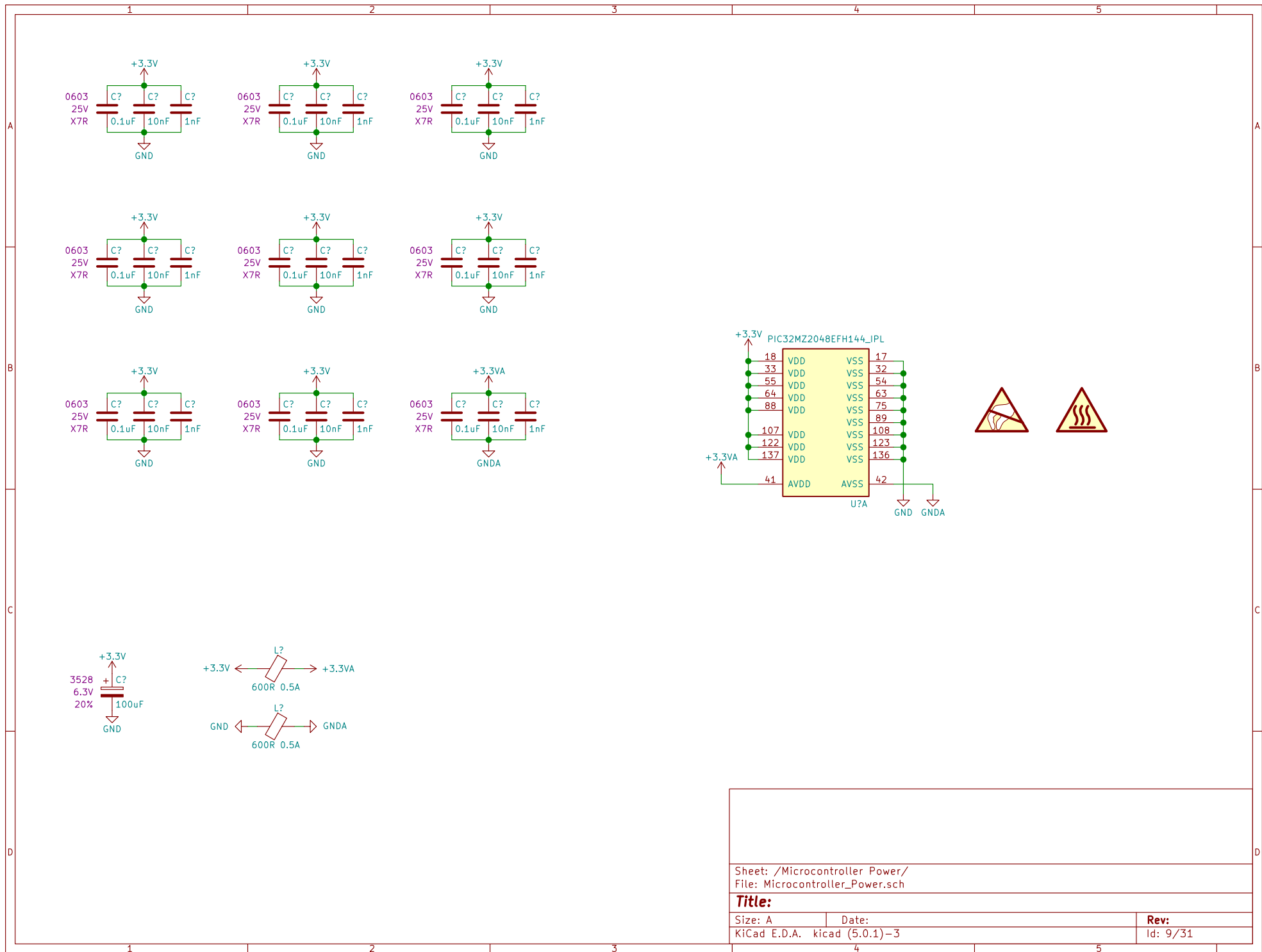
Date:

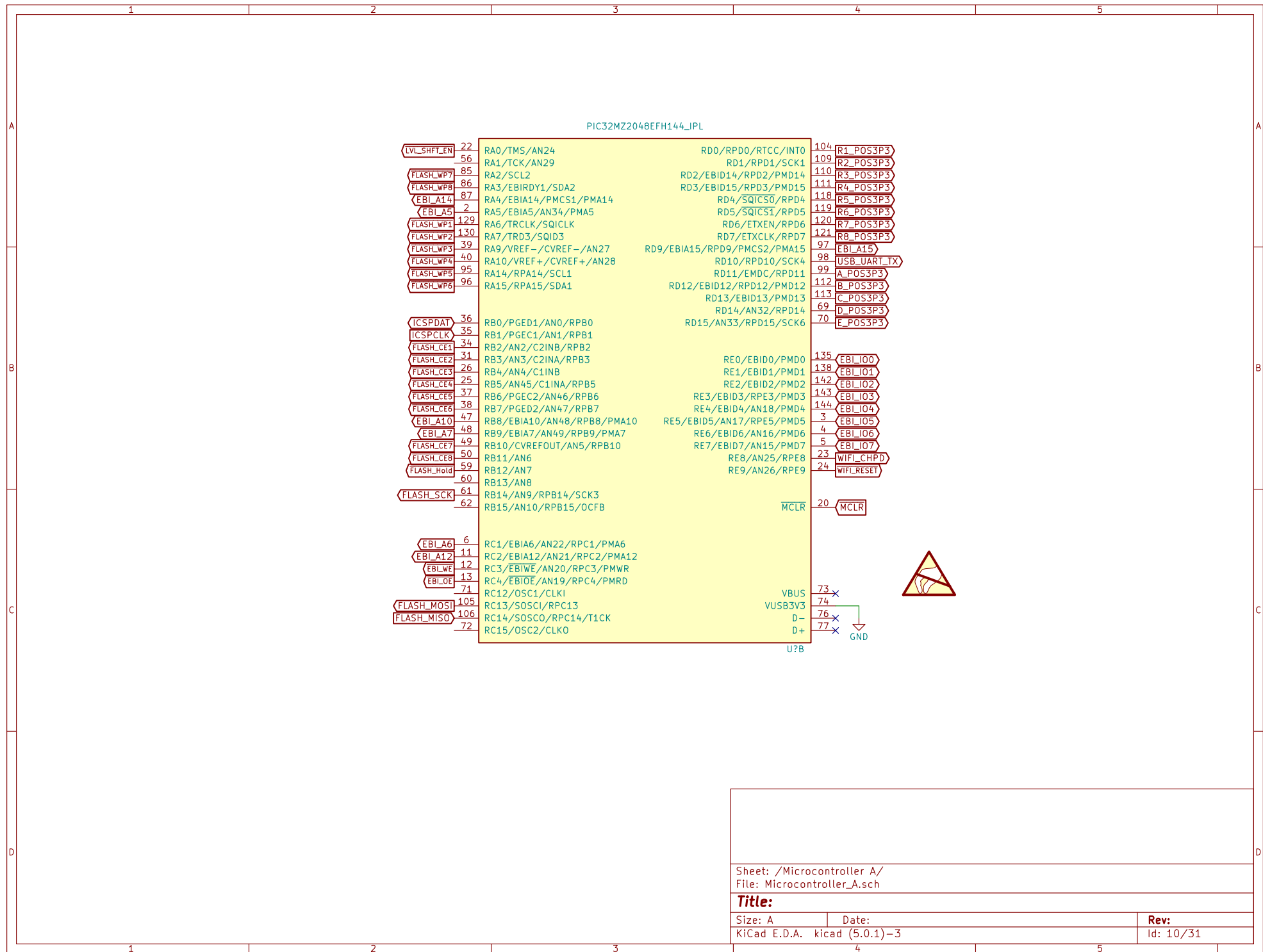
KiCad E.D.A. kicad (5.0.1)-3

Rev:

Id: 6/31







Sheet: /Microcontroller A/
File: Microcontroller_A.sch

Title:

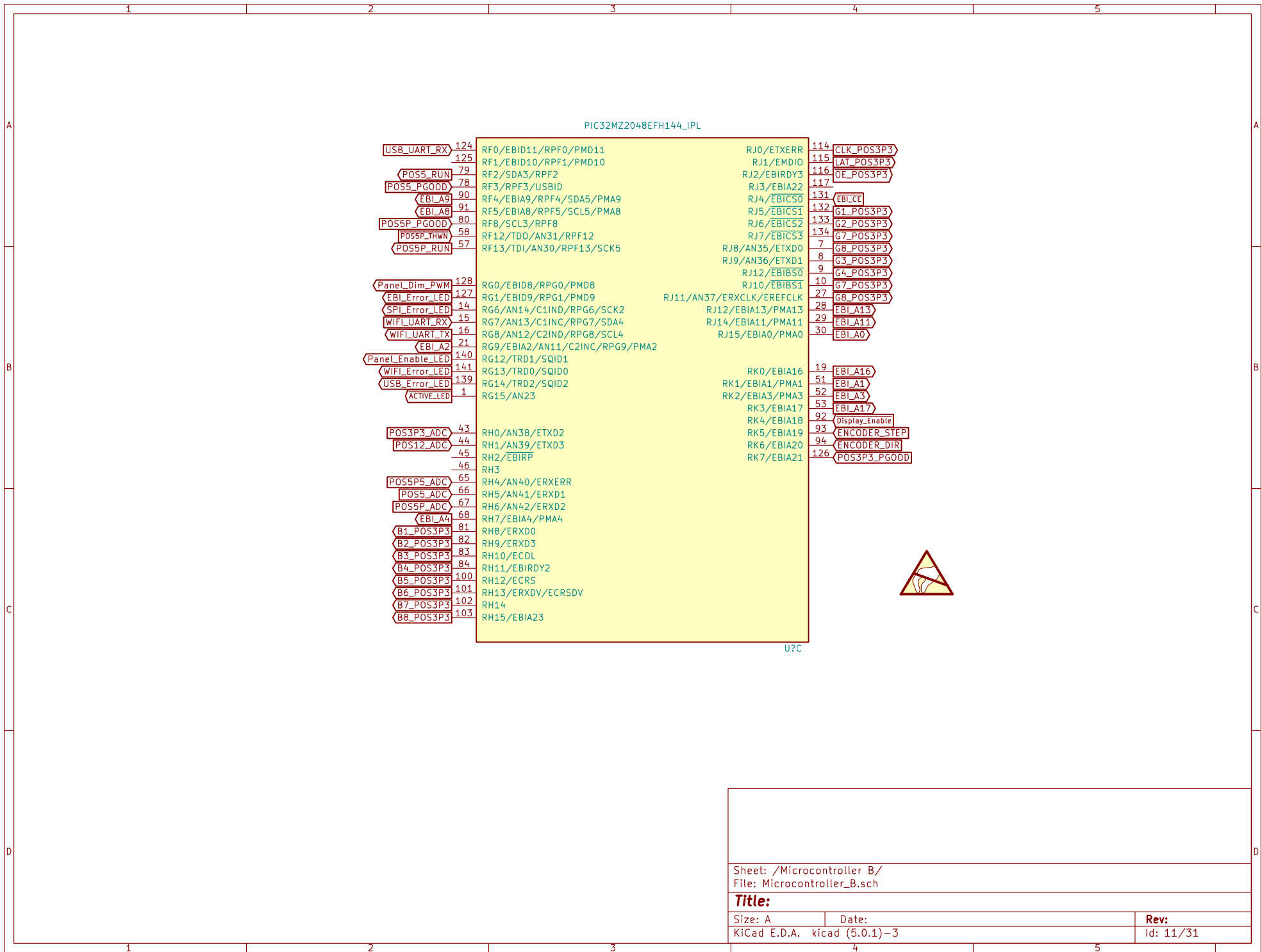
Size: A

Date:

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

Rev:

Id: 10/31



Sheet: /Microcontroller B/
File: Microcontroller_B.sch

Title:

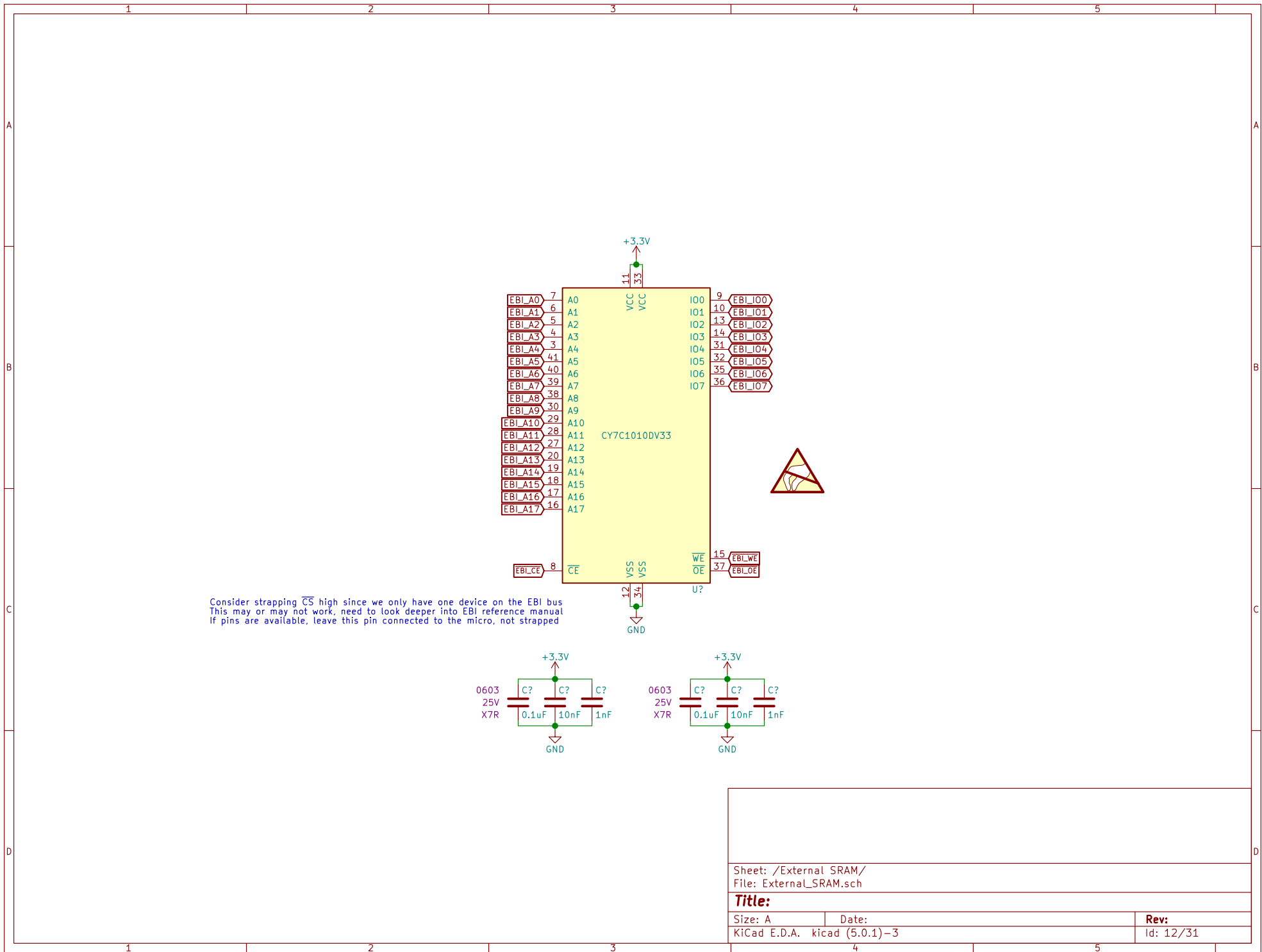
Size: A

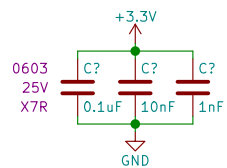
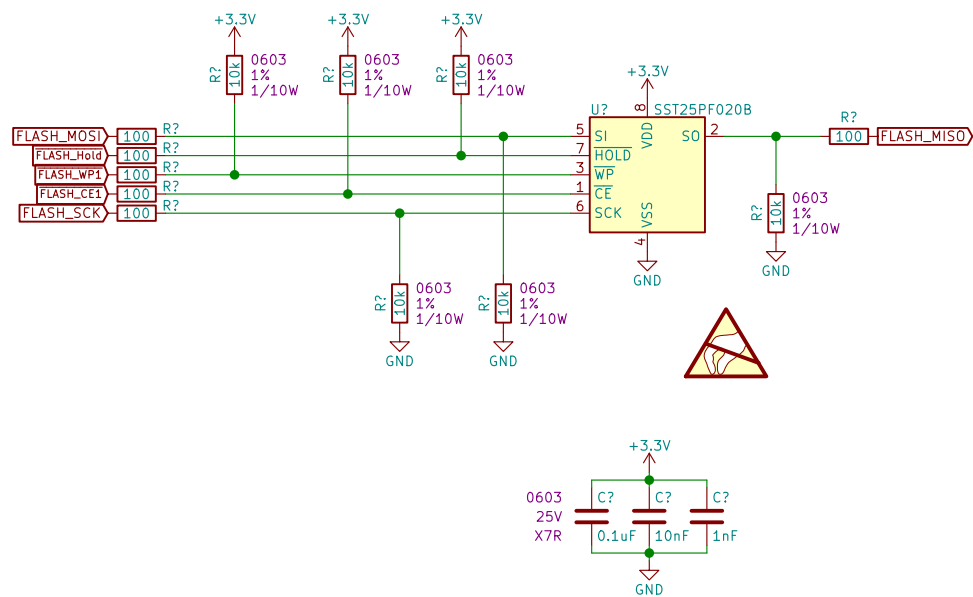
Date:

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

Rev:

Id: 11/31





Sheet: /External Flash 1/
File: External_Flash_1.sch

Title:

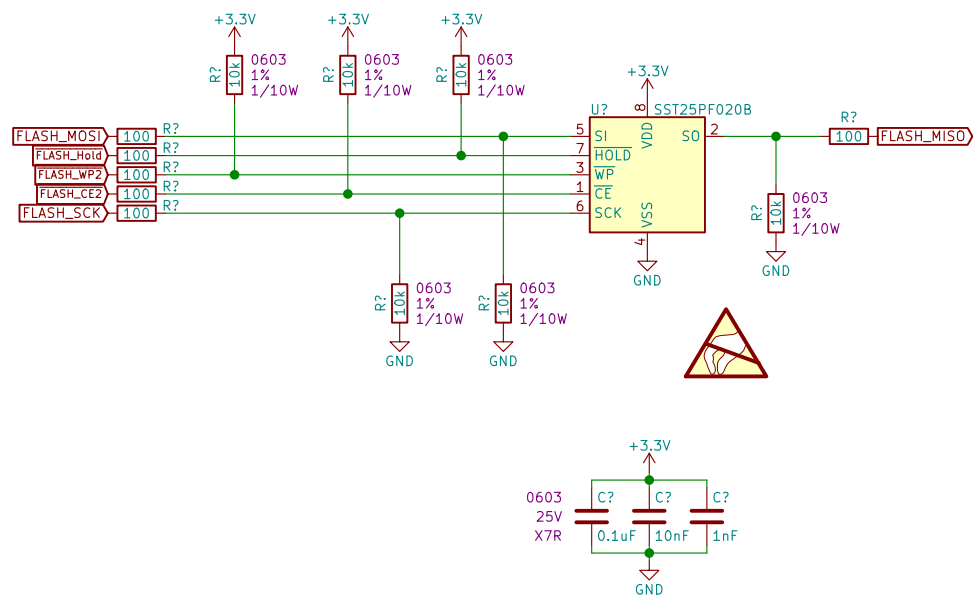
Size: A

Date:

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

Rev:

Id: 13/31



Sheet: /External Flash 2/
File: External_Flash_2.sch

Title:

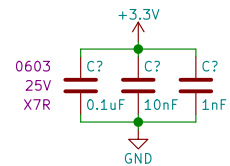
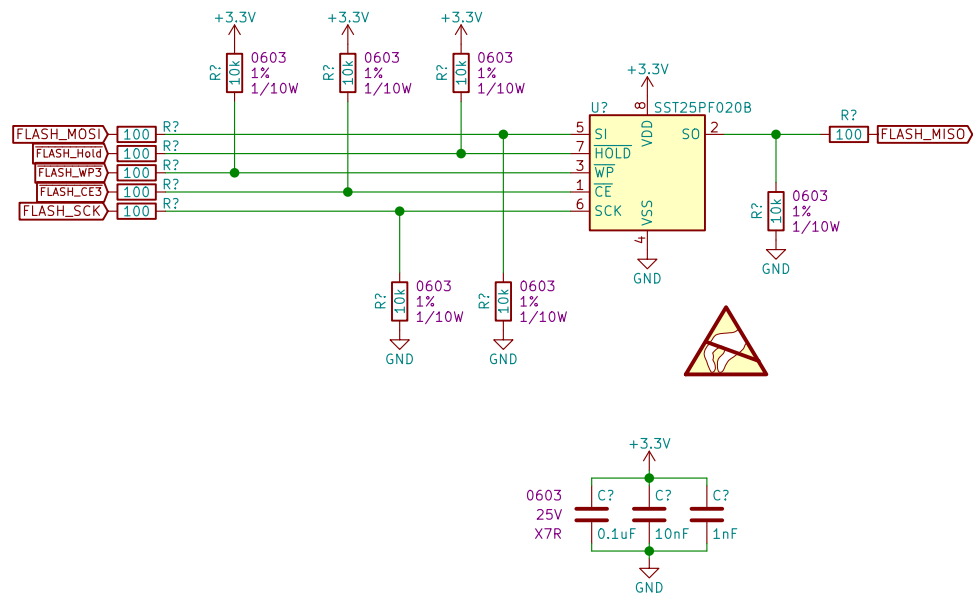
Size: A

Date:

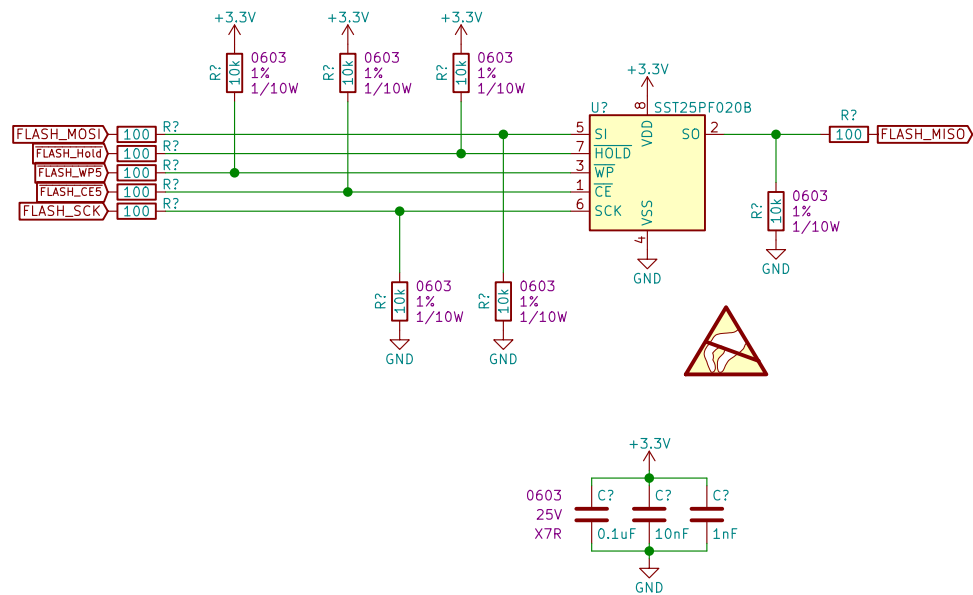
KiCad E.D.A. kicad (5.0.1)-3

Rev:

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 5/
File: External_Flash_5.sch

Title:

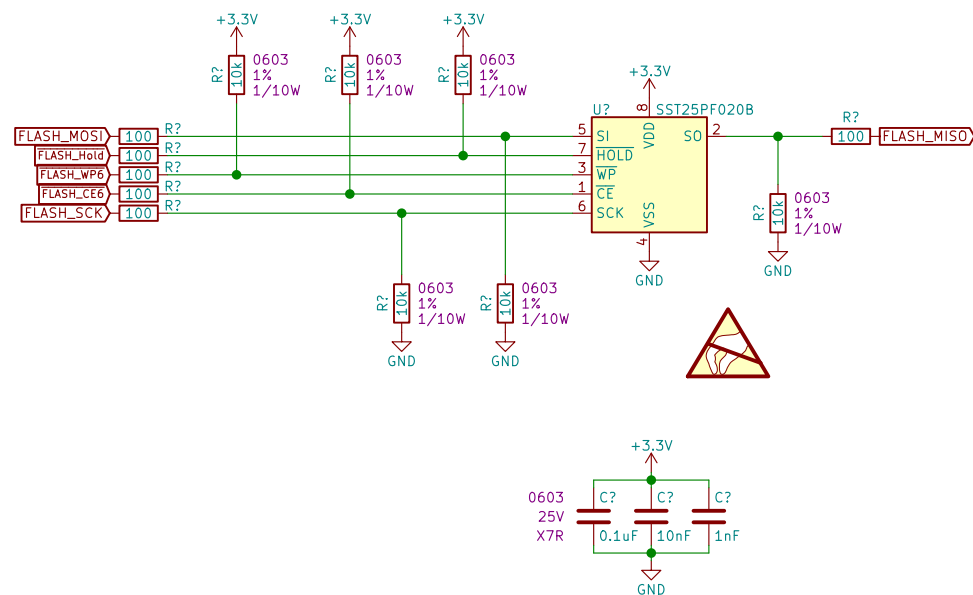
Size: A

Date:

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

Rev:

Id: 17/31



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

Title:

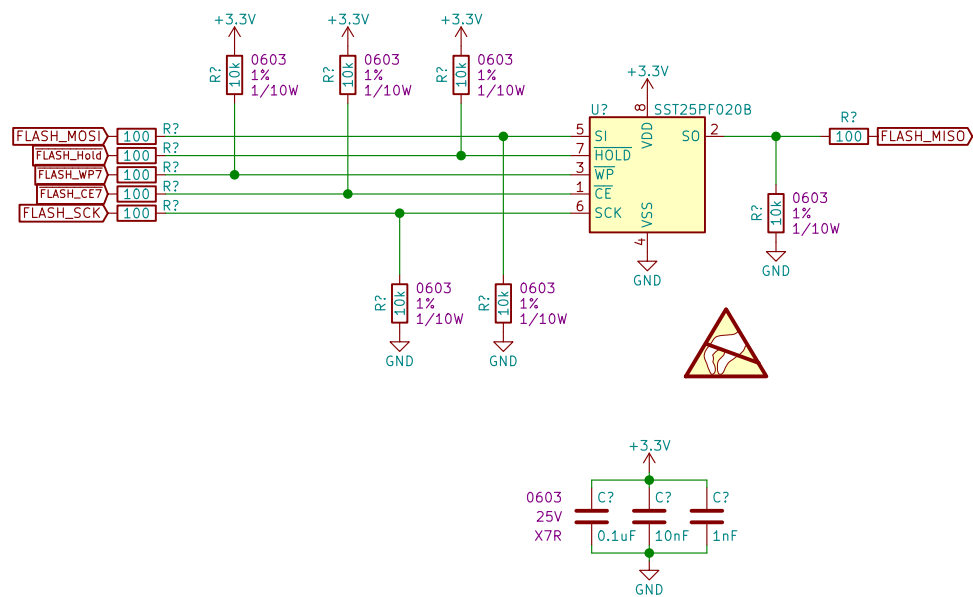
Size: A

Date:

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

Rev:

Id: 18/31



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

Title:

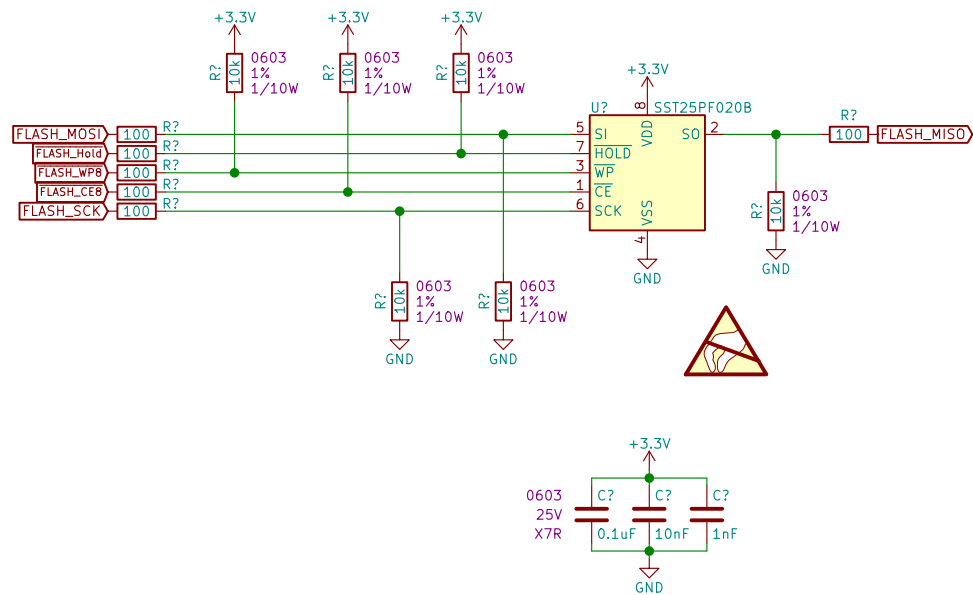
Size: A

Date:

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

Rev:

Id: 19/31



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

Title:

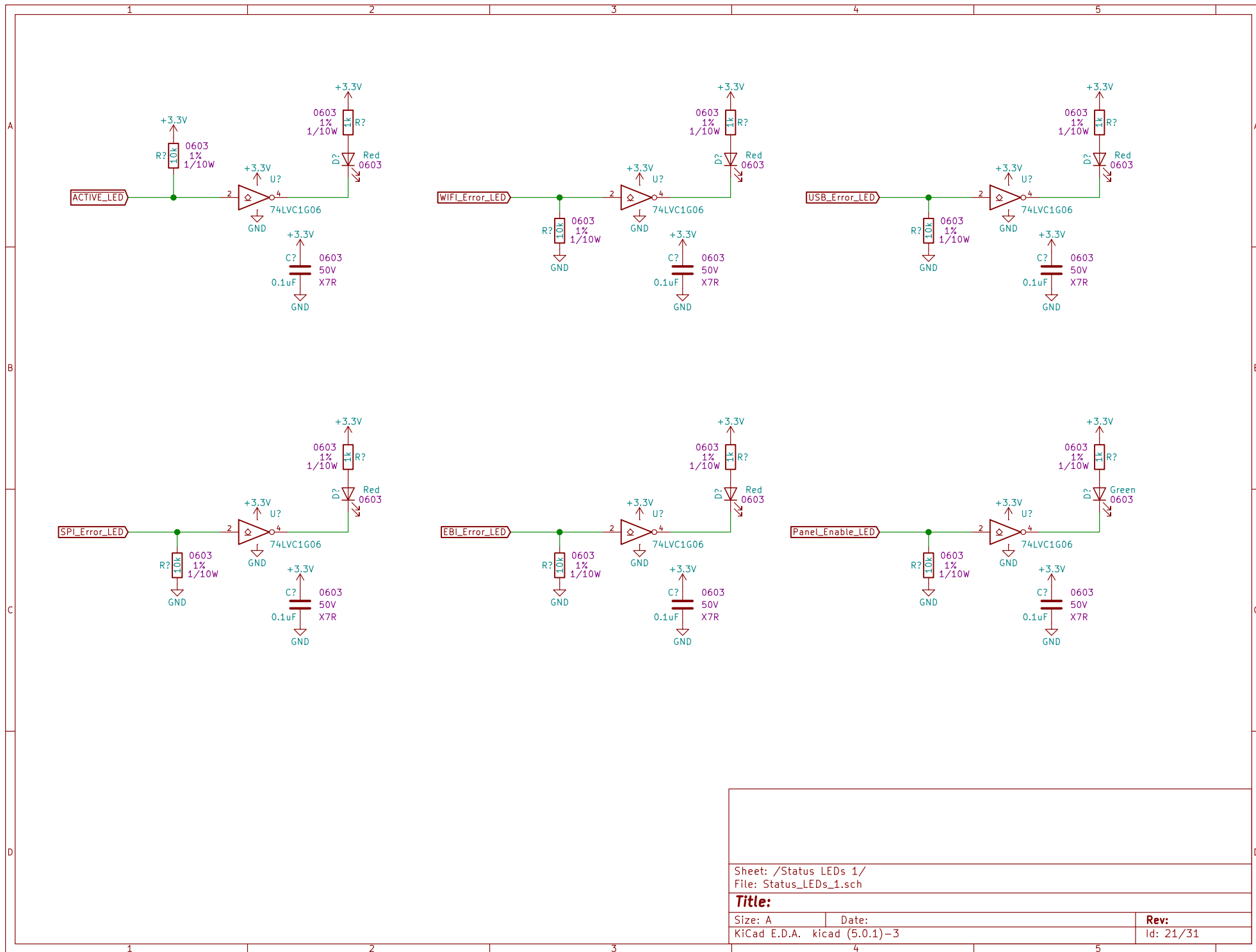
Size: A

Date:

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

Rev:

Id: 20/31

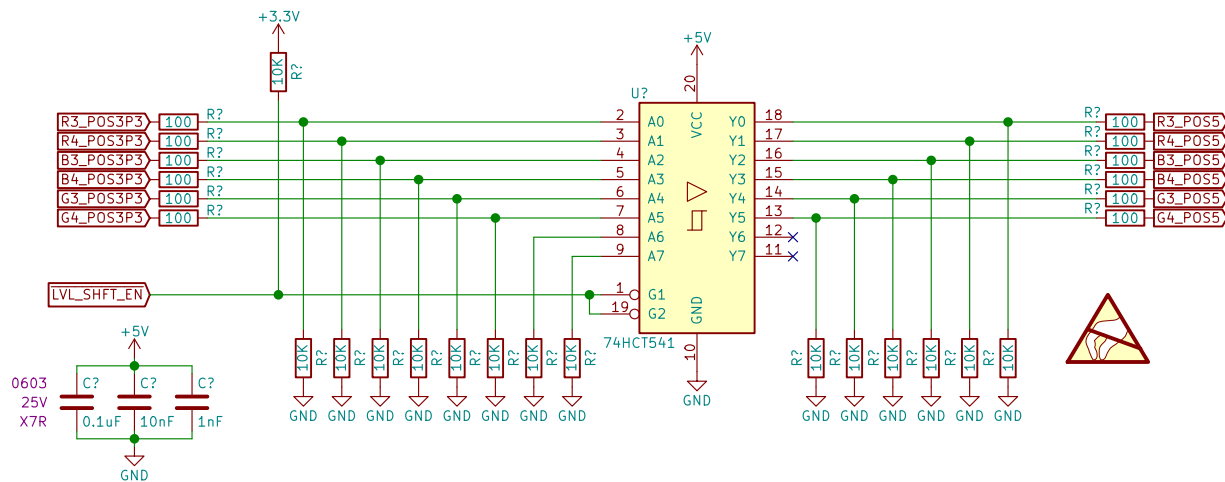
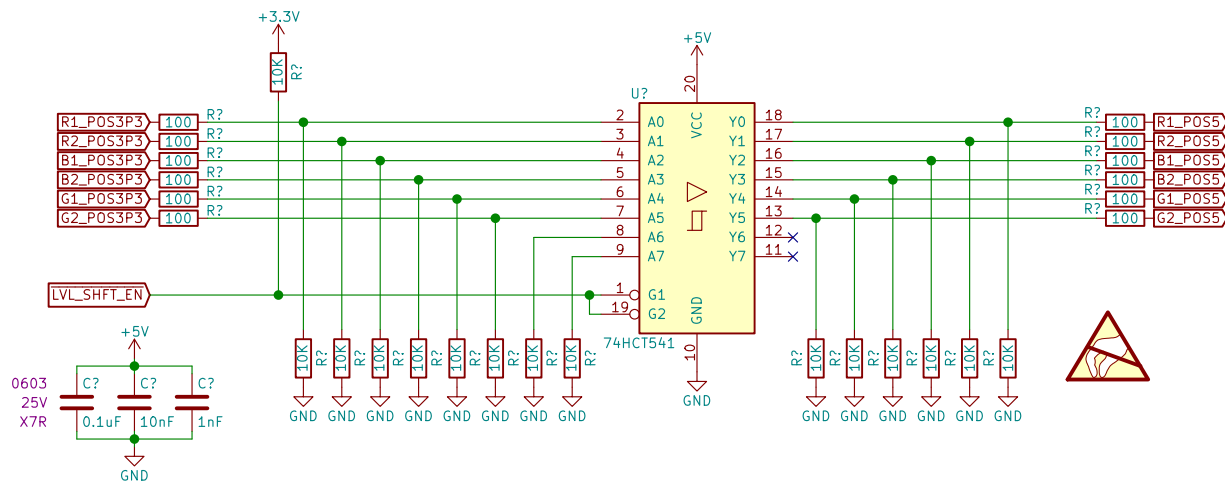


Sheet: /Status LEDs 1/
File: Status_LEDs_1.sch

Title:

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

Rev:
Id: 21/31

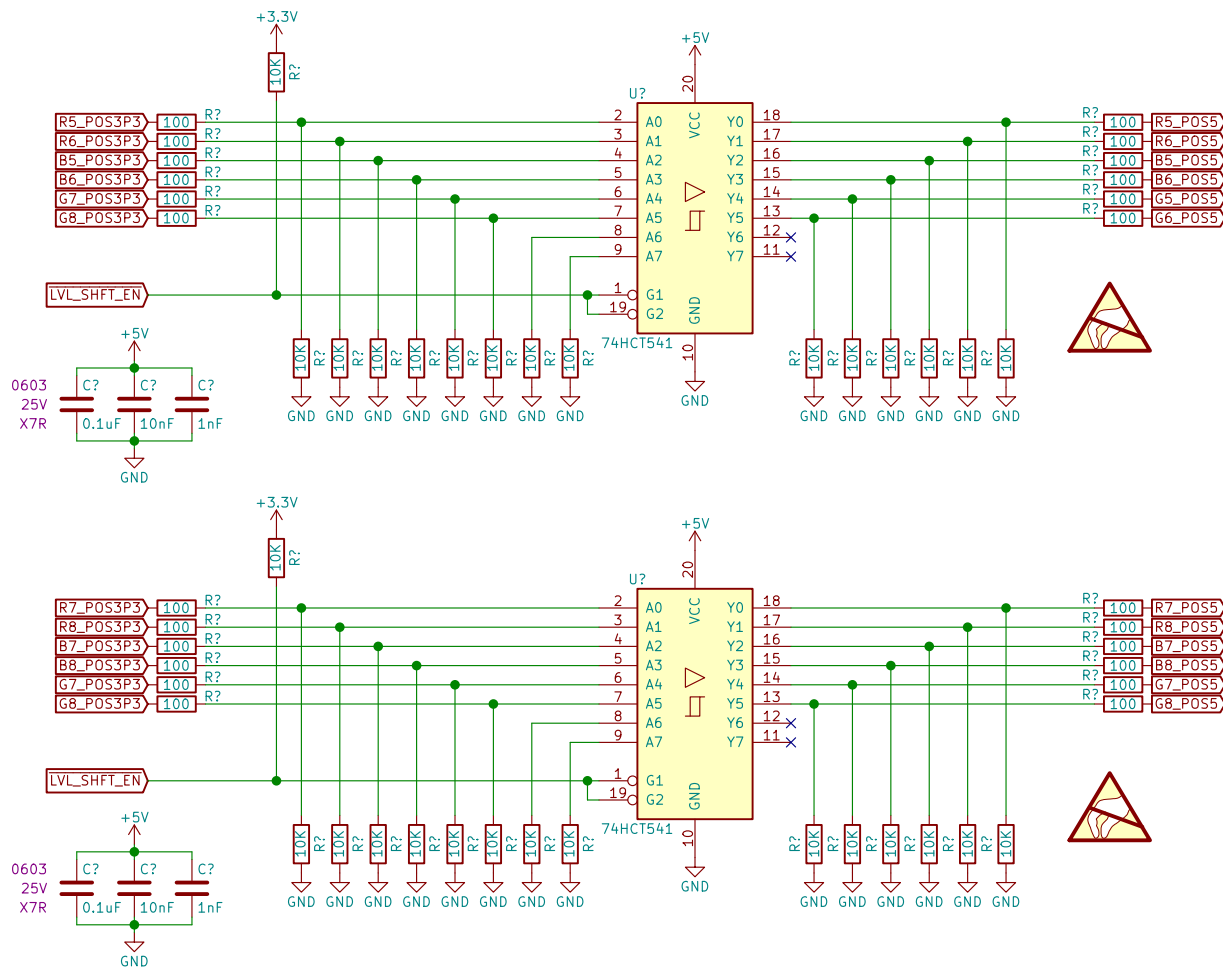


Sheet: /Panel Data Level Shifters 1/
File: PanelData_LevelShifters_1.sch

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:

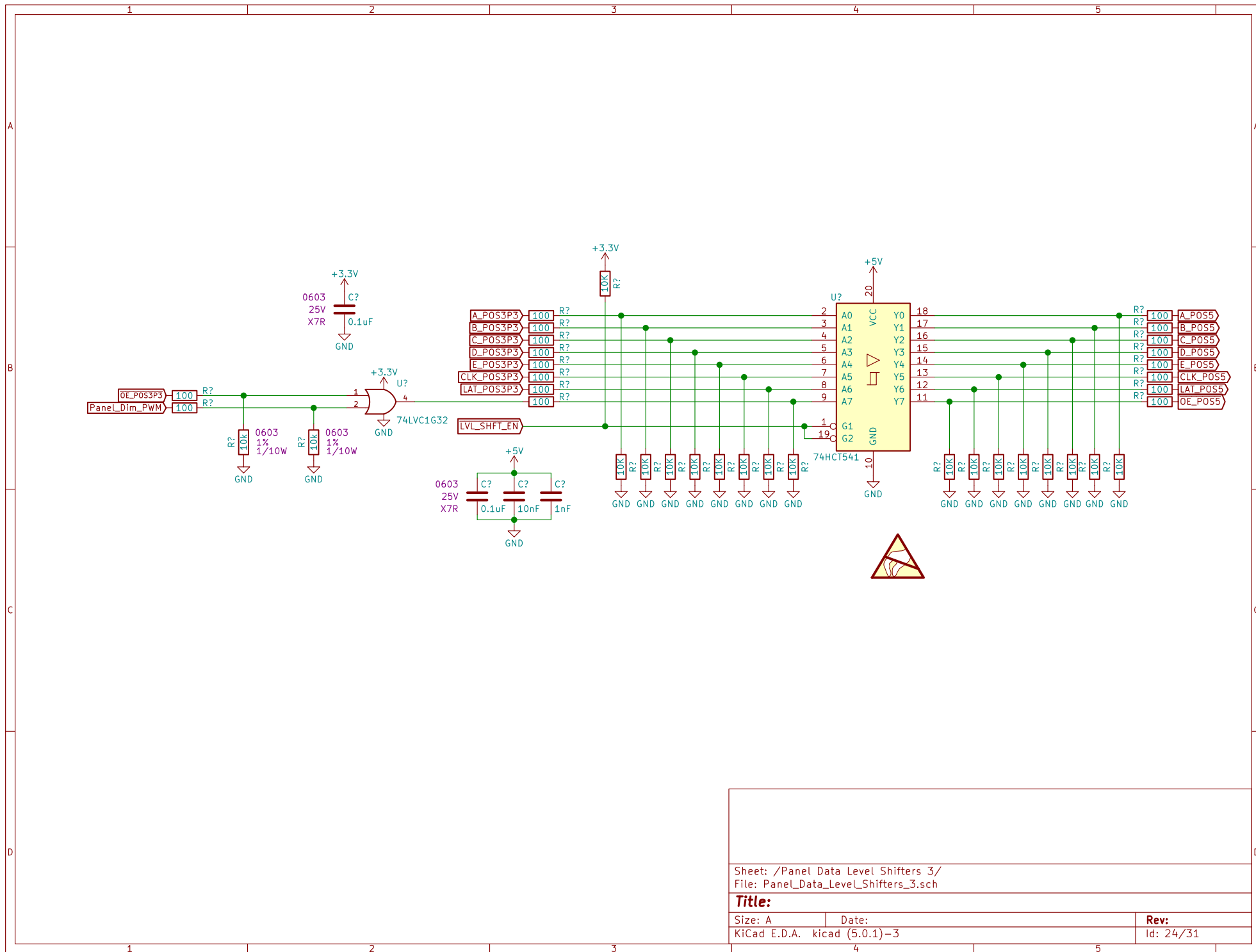
Size: A

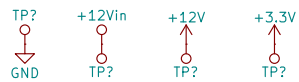
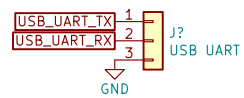
Date:

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

Rev:

Id: 23/31





Sheet: /Test Points/
File: Test_Points.sch

Title:

Size: A

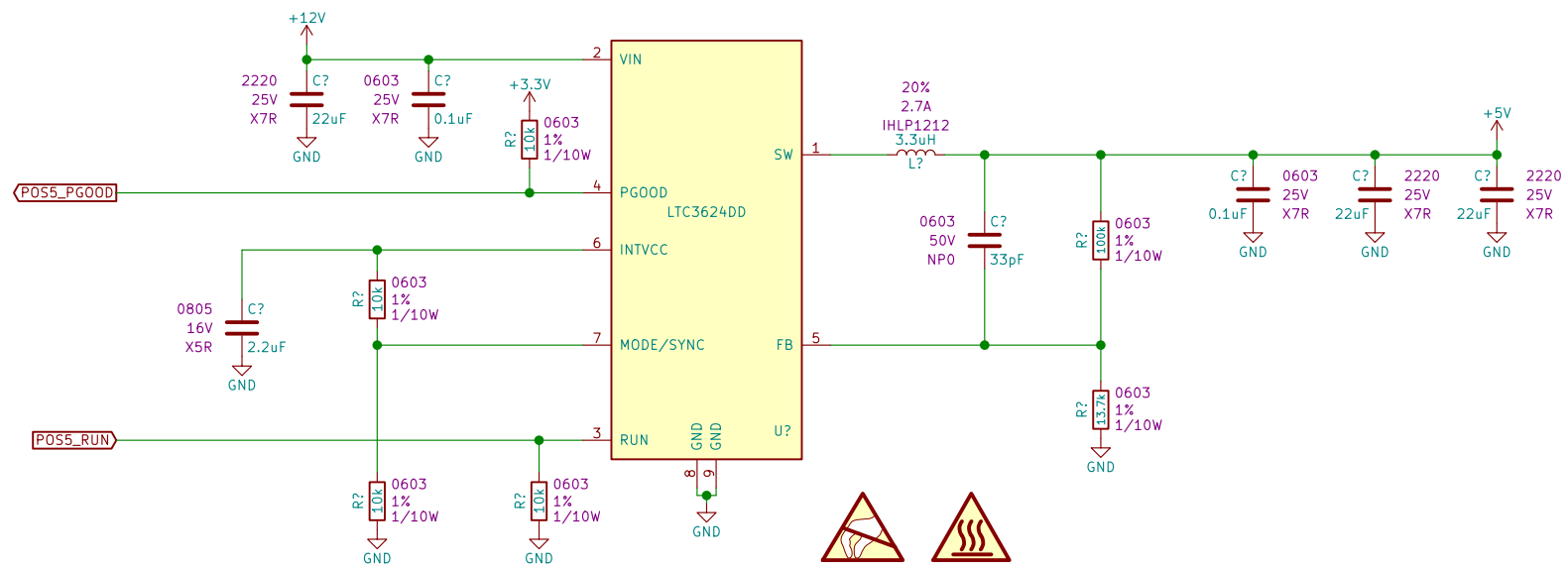
Date:

Size: 7	Date:
KiCad E.D.A.	kicad (5.0.1)-3

Rev:

Id: 25/31





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

Title:

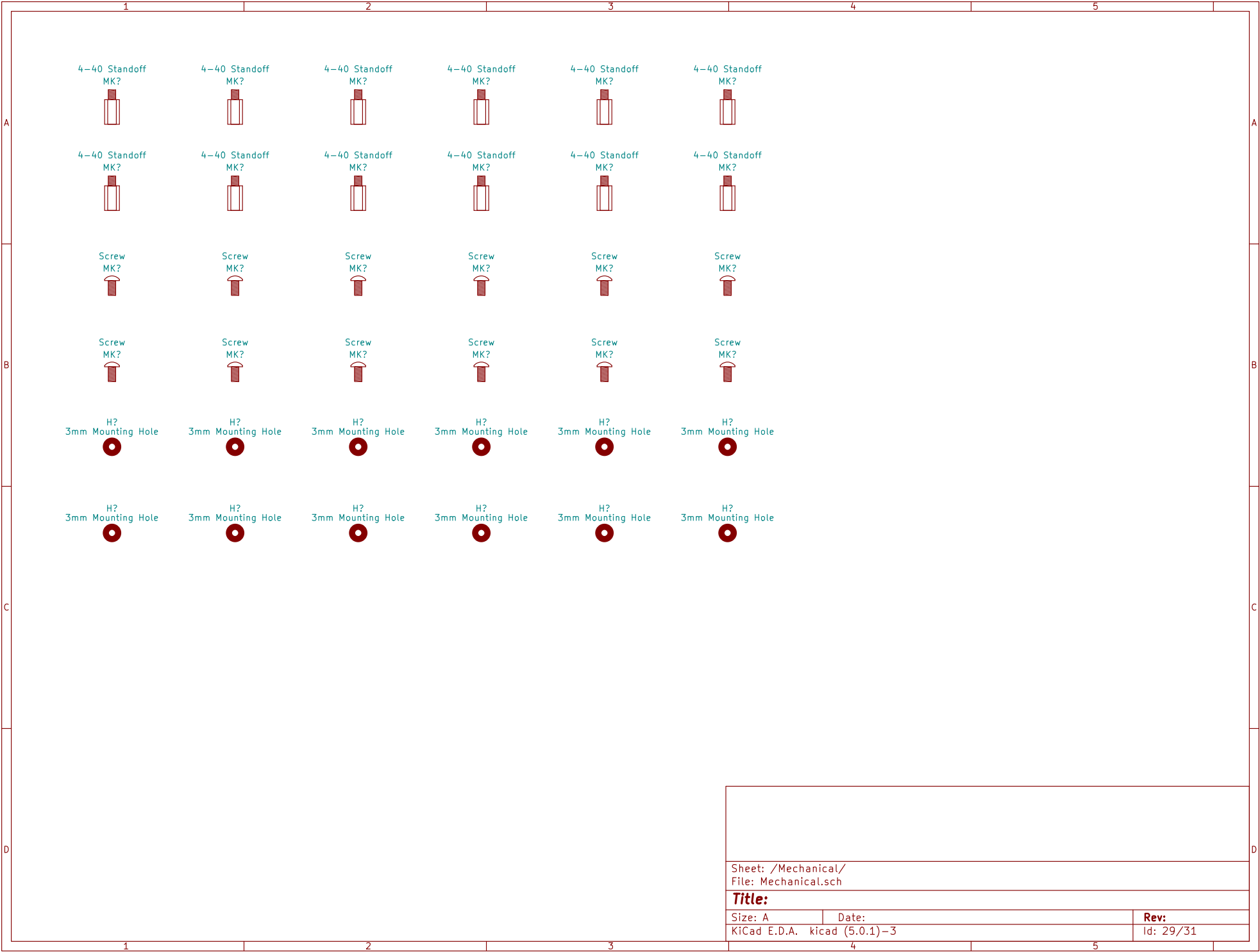
Size: A

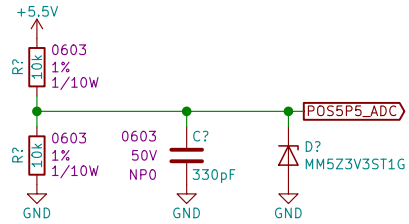
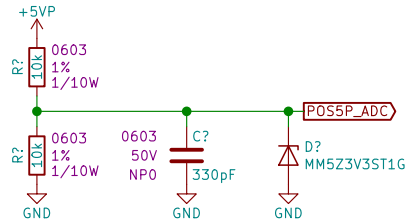
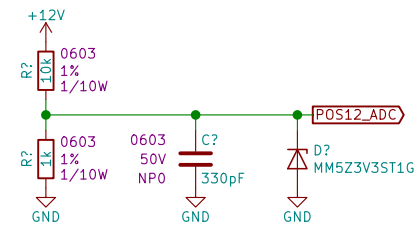
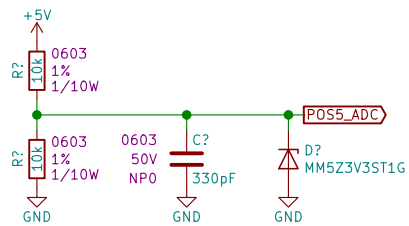
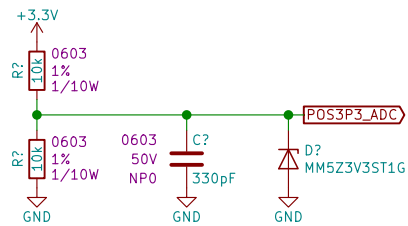
Date:

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

Rev:

Id: 28/31





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

