

This is the interface of the driver board.

- The user is expected to supply 3.3V and GND via this header.
- The commands can then be send via SPI to the MCU.

EPD Power Rails

+3.3V
 I2C1_SDA → 3.3VIN
 I2C1_SCL → SDA
 TPS_WAKEUP → WAKEUP
 TPS_PWRUP → PWRUP
 TPS_VCOM_CTRL → VCOM_CTRL
 TPS_PWARGOOD → PWARGOOD
 TPS_INT → INT
 -20V
 -15V
 +22V
 +15V
 VCOM
 +3.3V_EDP
 3.3V EPD

File: power_management.kicad_sch

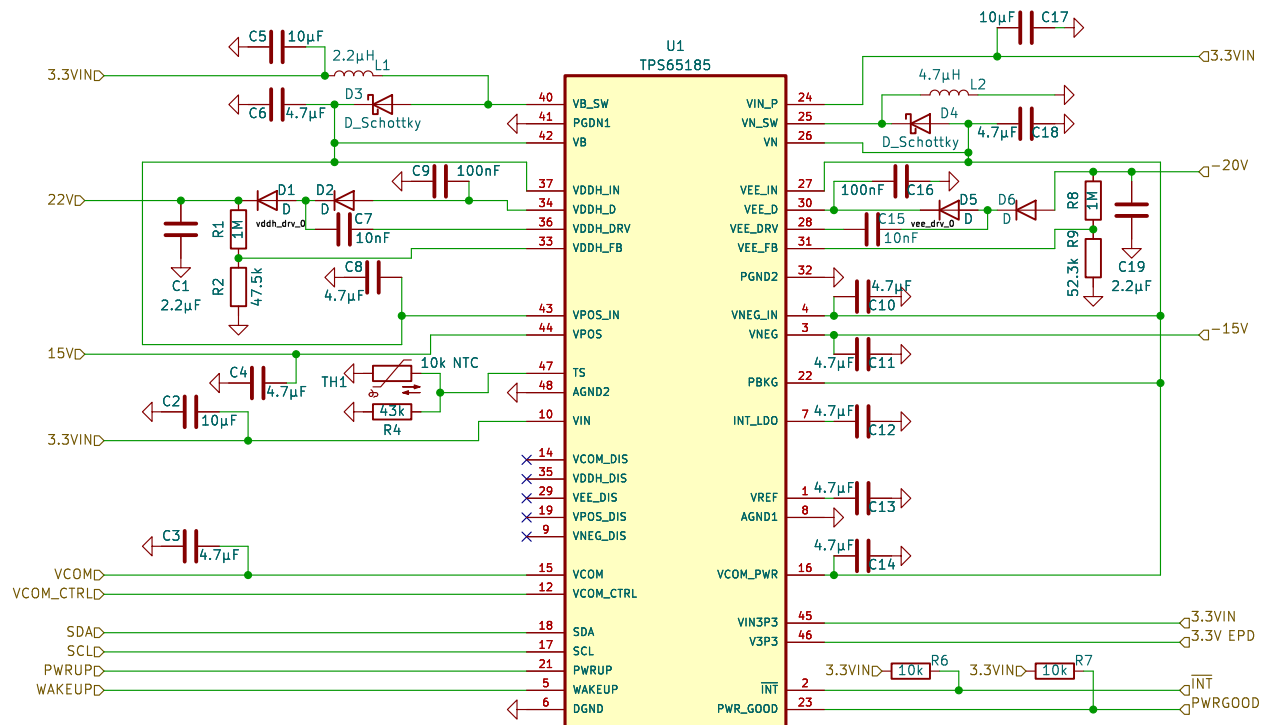
[illegible]

Diagram illustrating the pin configuration for the EPD ED060SC4 module. The module is connected to a microcontroller (MCU) via a 16-pin connector. The pins are labeled as follows:

- Power Pins:**
 - +3.3V_EDP (Green)
 - VCOM (Green)
 - +15V (Red)
 - +22V (Red)
 - 15V (Green)
 - 20V (Green)
- Data Pins:**
 - D0 (Green)
 - D1 (Green)
 - D2 (Green)
 - D3 (Green)
 - D4 (Green)
 - D5 (Green)
 - D6 (Green)
 - D7 (Green)
- Control Pins:**
 - EPD_OE (Green)
 - EPD_LE (Green)
 - EPD_SPH (Green)
 - EPD_CLK (Green)
 - EPD_GMODE (Green)
 - EPD_CKV (Green)
 - EPD_SPV (Green)

The file path for the schematic is: `file: epd_ed060sc4.kicad_sch`

Id: 1/3



This schematic is based on the "Typical Application" of the TPS65185 datasheet in Section 9.2. It is suited for the required rails -20V, -15V, 15V, 22V and 3.3V.

This schematic assumes a VIN power of 3.3V, that is also used for the pull-up resistors of $\overline{\text{INT}}$ and PWR_GOOD.

Sheet: /EPD Power Rails/
File: power_management.kicad_sch

Title:

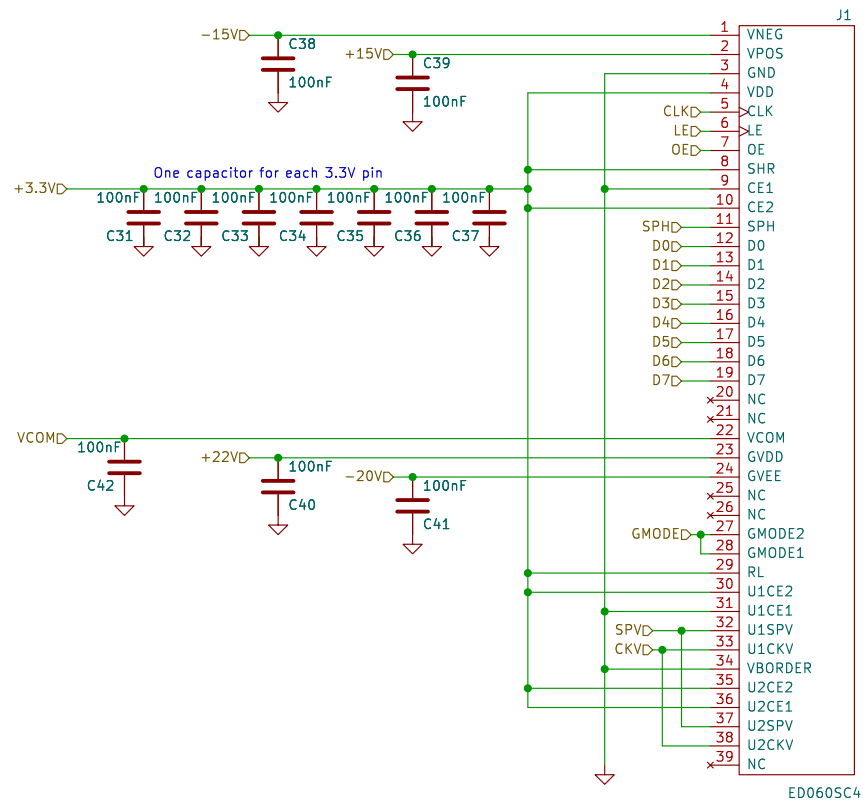
Size: A4

Date:

Rev: 0.1

KiCad E.D.A. kicad (7.0.0)

Id: 2/3



100nF capacitors should be OK everywhere.
I got no data whatsoever to support that claim though.

Sheet: /EPD ED060SC4/
File: epd_ed060sc4.kicad_sch

Title:

Size: A4
KiCad E.D.A. kicad (7.0.0)

Date:

Rev:
Id: 3/3