PRODUCT DATASHEET

Scoop Datasheet

The Scoop provides backup power and facilitates reliable power management for Notecard-powered IoT applications. The Scoop includes a Lithium-Ion Capacitor (LIC), a hybrid type of capacitor classified as a type of supercapacitor.

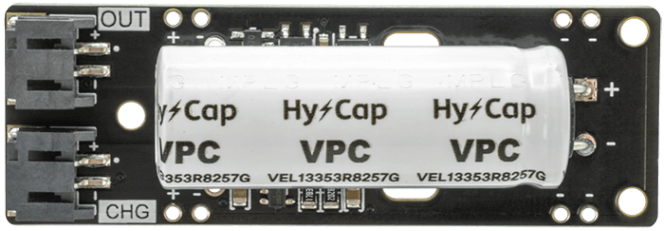
Blues Scoop™ Datasheet

Functional Description

The Scoop provides backup power and facilitates reliable power management for Notecard-powered IoT applications. The Scoop includes a Lithium-Ion Capacitor (LIC), a hybrid type of capacitor classified as a type of supercapacitor. LICs have numerous advantages in comparison to using Lithium Polymer (LiPo) batteries as a power source as, in general, they are more reliable, have the ability to operate in a wider range of temperatures, experience a lower self-discharge rate, and are significantly safer to ship.

The Scoop includes a 250 Farads LIC, two JST connectors, and four pairs of header pins for flexible connectivity.

View in Store

Front	Back
	

Features

- **Capacitor:** Lithium-Ion 250F 3.8V. Provides reliable power and stable delivery over a short period of time.
- **Connectors:** JST Battery Connectors PH-2POS wired for Adafruit-style packs.
- **Charge Connector:** J1, Accepts 4.8V-24V (see Board Errata below).
- **Output Connector:** J2, Outputs 2.5V-3.8V.
- **Header Pins:** 4 pairs of through-hole mounted male header pins.

> **Note**
>
> Based on internal testing, Scoop is able to power up to approximately 20 on/off cellular connection cycles, or up to 240 minutes of a single cellular connection. Results may vary depending the type of wireless coverage in the area and the amount and frequency of data being sent.

Power Information

The Scoop can be powered via its charge connector (`CHG`), which accepts an input voltage of 4.8V-24V (see Board Errata below), or via the adjacent header pins.

The output (`OUT`) connector can deliver a voltage in the range of 2.5V-3.8V.

Dimensions

The Scoop was designed with a minimal footprint for easy embedding in existing applications.

- 59.69mm (2.35 inches) wide
- 20.0mm (0.79 inches) tall
- 14.16mm (0.56 inches) deep

Header Pins

The Scoop features four pairs of header pins, labeled as follows:

- J3 header pair (next to `CHG`): Power source
- J4 header pair (next to `OUT`): Load circuit

Board Errata

Serigraphy on the Scoop lists the input charge voltage as 4.8V-24V. However, if Scoop is connected to a Notecarrier F and is fully discharged, when a voltage >5V is subsequently applied, the LIC on the Scoop will not charge well. Additionally, the Notecard and host MCU attached to the Notecarrier F will not be able to run and may enter a boot loop.

This is due to a current draw limitation on the power converter on the Scoop that limits the current depending on the voltage differential across the power converter. For higher voltage levels, the input current draw is lowered enough so that the LIC takes a long time to charge, and will keep getting discharged by the Notecard while it attempts to power on.

Design Files

Open source hardware designs for the Scoop and all Blues hardware are maintained in the note-hardware GitHub repository.

Here are the design files available for the Scoop:

- Schematic (`.pdf`) — the full circuit schematic. Use it to understand how the lithium-ion capacitor, charge control, and output switching are wired together when designing a compatible host or debugging a connection.
- Bill of materials (`.xlsx`) — every component with its reference designator and manufacturer part number. Use it to source parts or build your own board.
- Overview drawing (`.pdf`) — a dimensioned drawing of the board outline, mounting-hole locations, and connector positions for reference and mechanical fit.
- Gerber fabrication files (`.zip`) — the Gerber and drill set. Send this package to a PCB manufacturer to have bare boards produced.
- 3D model (`.step`) — a STEP 3D model. Import it into mechanical CAD to check fit and design enclosures.
- Kapton tape drawing (`.pdf`) — shows where insulating tape is applied during assembly.
- KiCad project (folder) — the complete, editable KiCad design (schematic, PCB layout, and libraries). Open it in KiCad to modify the design or generate your own outputs.

Can we improve this page? Send us feedback