

An Alternate Organization System for I²C Target Devices to Increase Accessibility and Efficiency

And a Breakout Board Utilizing the New System

Pranit Kumar

August, 2025

Abstract

This project resulted in the creation of an accessible solution to the headache of wiring multiple I²C Devices. It allows many I²C Devices to easily be connected at the same time, while simultaneously being easy to physically add or remove. This new solution culminated in the development of a physical breakout board that allows up to six target devices to be connected to the controller device, with options to add power regulation.

1 Introduction

1.1 Background

The Inter-Integrated Circuit (I²C) is a communication protocol formulated by NXP Semiconductors, previously a division of Philips, in 1982. It was designed to enable straightforward communication among low-speed integrated circuits. Today, it has extensive utilization across a broad spectrum, encompassing hobbyists, enthusiasts, and industrial applications. I²C uses a total of four pins, those being VCC (Power), GND (Ground), SDA (Serial Data), and SCL (Serial Clock). Only two of those four connections actually make up the I²C transmission— SDA and SCL. The SDA line has two way communication, allowing the controller device and target device to freely speak to each other. The SDA line is the data transmission line, using high and low to convey bits. The SCL line synchronizes every device on the I²C Bus with regular clock pulses coming from the controller device.

1.2 The Problem

I²C works great in many different situations— from consumer electronics such as cell phones and computers, to connecting sensors to an Arduino. However, there is a significant issue that arises only in the latter, which is the annoyance that routing the target devices to the controllers is. Traditionally, the jumper cables sprouting from the controller, such as a Raspberry Pi, Arduino, or other I²C enabled controller devices, are plugged into a breadboard in a group of four pins, going down the columns. This creates a row for each of the four pins. Target devices have their pins plugged into the rows that have been created. The wires connecting to devices sprout up and out of the breadboard, as shown in Figure 1. This limits movement of the target device parallel to the pins, while movement perpendicular to the pins remains easily accessible.

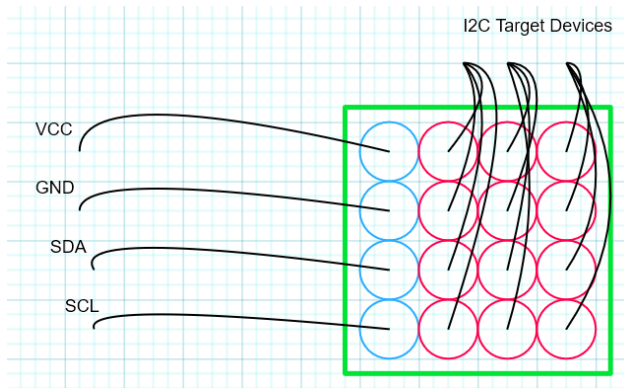


Figure 1: A Traditional Method of Wiring I²C Devices

This creates a headache for anyone wiring together all of the targets to the controller and breadboard, since they have to allow devices to fall in the same direction. When the targets fall in the same direction, their cables tend to get tangled together, which makes removing cables much more difficult than it should have to be. In addition, if you skip a column for any reason, and need to later add a target to those pins, you will have to reach

through the jumper cables of other devices to add more in. This problem does not occur in more industrial uses of I²C, where cables and breadboards are not used in favor of custom fabricated PCBs with traces. Because traces are used rather than cables, and the controller and targets are all soldered to boards, the issue of human inaccessibility does not exist. This unorganized method of I²C cable management and connection is what causes it to be annoying to work with.

2 Proposed Solution

2.1 Theory

The fix to this annoying and inaccessible system is by grouping the I²C pins together, rather than having them in rows like before. Each set of four pins is grouped into a square, kept at a reasonable distance from the other groups, to ensure usability. This arrangement allows target devices to be plugged in and removed from any direction, since no cable is forced to travel parallel to another in order to reach its pin. The result is a setup where adding, removing, or repositioning a target device is a matter of seconds rather than a frustrating untangling exercise.

2.2 Initial Application: Proof of Concept during NASA TechRise

The first real-world test of this grouping approach came during the NASA TechRise Student Challenge, a competition in which student teams design, build, and fly a suborbital science payload aboard a high-altitude balloon or rocket platform. The payload built for this competition required several I²C sensors to be connected simultaneously to a single microcontroller—a scenario that made the traditional wiring method’s shortcomings immediately apparent. Inside a payload enclosure, space is at a premium, and tangled or inaccessible cables are not merely annoying; they are a genuine engineering liability that can prevent last-minute adjustments before flight.

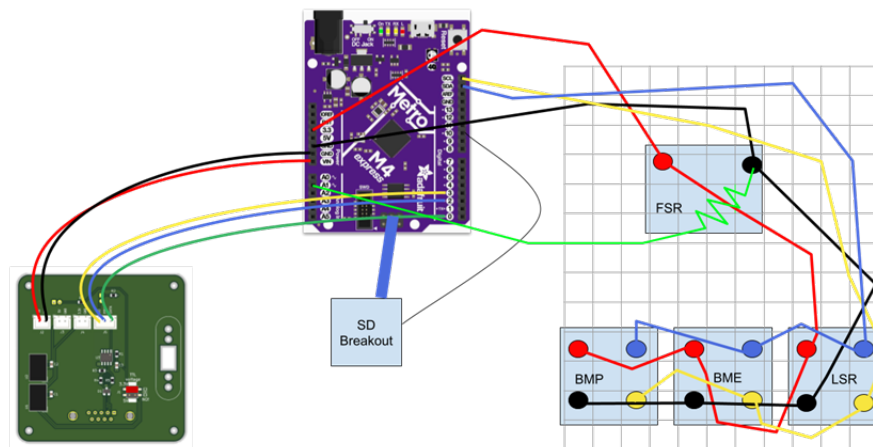


Figure 2: A Rudimentary Form of this Proposal, as Applied in the NASA TechRise Payload

To address this, the four I²C pins for each target device were grouped into a square cluster on a small perfboard, as shown in Figure 3. Each cluster was spaced far enough from its neighbors that a connector could be seated or removed without disturbing adjacent connections. While this implementation was rudimentary—hand-wired and lacking the refinements of a dedicated PCB—it validated the core concept convincingly. Sensors could be swapped in and out cleanly, cable routing inside the enclosure remained orderly, and the team was able to make configuration changes in the final hours before payload submission without any of the frustration that the traditional row-based approach would have introduced. That successful flight demonstrated that the grouped-pin organization was worth formalizing into a purpose-built board.

3 Breakout Board

3.1 Why is a Custom PCB Necessary?

The perfboard implementation used during NASA TechRise proved the concept works, but it came with the limitations inherent to any hand-wired prototype. Traces soldered by hand vary in resistance and reliability, exposed connections are vulnerable to shorts, and reproducing the same board a second time requires redoing all of the same work from scratch. A custom Printed Circuit Board (PCB) eliminates all of these problems at once. Traces are etched to exact tolerances, every copy of the board is identical, and the finished product is compact and robust enough to be used repeatedly without concern. Beyond reliability, a custom PCB also enables features that are impractical to implement on perfboard—specifically, integrated power regulation, which allows the board to supply the correct voltage to target devices directly rather than requiring the user to manage power separately. For a solution intended to be genuinely useful rather than merely functional, a PCB is not an optional upgrade; it is the logical conclusion of the design.

3.2 Design Requirements

Before beginning the design, a clear set of requirements was established to ensure the final board would actually solve the problem it was built to address. First, the board needed to support a minimum of six simultaneous I²C target device connections, as this covers the vast majority of hobbyist and educational use cases. Second, each connection group had to follow the square-cluster pin arrangement described in Section 2, with sufficient physical spacing between clusters to allow connectors to be seated and removed without obstruction. Third, the board required optional onboard power regulation, giving users the choice to supply regulated power to their target devices directly from the board rather than through a separate circuit. Fourth, the board had to maintain full compatibility with common controller devices, including the Arduino Uno, Arduino Nano, and Raspberry Pi, without requiring any adaptation or modification on the controller side. Finally, the overall form factor needed to remain compact enough to be practical inside a project enclosure or mounted alongside other components in a confined space.

3.3 Designing Phase

The board was designed using *KiCAD 9.0.2*, an open-source PCB design suite that integrates schematic capture, board layout, design-rule verification, and Gerber export into a single workflow. Unlike the earlier perfboard prototype used during NASA TechRise, this version was intended to be compact, reproducible, and mechanically reliable enough for repeated real-world use. The final board dimensions were approximately 54.5 mm \times 29.0 mm, with a standard thickness of 1.6 mm and a two-layer copper stackup consisting of front copper (F.Cu) and back copper (B.Cu).

The schematic was intentionally kept electrically simple. The controller-side interface consists of four separate inputs: VCC, GND, SCL (Serial Clock), and SDA (Serial Data). Rather than combining these into a single keyed connector, the design uses individual through-hole connection points on the left side of the board. This decision was made to preserve compatibility with a wide variety of jumper-wire configurations commonly used with Arduino and Raspberry Pi systems.

These four nets are routed in parallel to six grouped target-device connectors labeled J1 through J6. Each connector is implemented as a 2 \times 2 through-hole pin header at a 2.54 mm pitch. This grouped square arrangement is the core feature of the design: instead of spreading the four I²C connections across breadboard rows, the four related pins are physically clustered together in a compact square. This allows cables to enter and exit from almost any direction without interfering with neighboring devices.

One of the more unusual aspects of the design is the deliberate omission of pull-up resistors on the SDA and SCL lines. In many beginner-oriented I²C breakout boards, pull-up resistors are permanently included on the PCB. However, many controller devices and sensor modules already contain onboard pull-ups internally. Adding additional pull-up resistors unnecessarily can reduce signal quality or increase current draw, especially when many target devices are connected simultaneously. Instead of forcing a fixed configuration, this board leaves pull-up implementation to the controller or target devices already present in the system. This design choice is explicitly labeled on the back silkscreen of the board to ensure users are aware of the requirement.

The PCB also includes several small surface-mount capacitors positioned throughout the design. Five 0.1 μ F bypass capacitors in 0603 packages were added to reduce high-frequency electrical noise on the power rails. In addition, a larger 10 μ F bulk capacitor in a 1210 package was included near the primary power distribution area to stabilize voltage during transient load changes. Although these components are physically small, they significantly improve the electrical stability of the board during operation, particularly when multiple sensors are connected simultaneously.

After the schematic was completed, the physical layout process began. The six connector groups were arranged in two rows of three, producing a compact rectangular footprint while still leaving enough spacing between connectors for accessibility. Care was taken to ensure that jumper wires and sensor cables could be connected or removed without obstructing adjacent ports. Most signal routing was completed on the front copper layer, while the back layer was used selectively to simplify trace crossings and reduce congestion.

The traces themselves were routed according to the manufacturing constraints established during the design process. The board was designed around a minimum trace width and

clearance of 0.127 mm, values compatible with inexpensive two-layer PCB fabrication services such as JLCPCB. Wider traces were used for the primary power distribution lines, while narrower traces were sufficient for the relatively low-current SDA and SCL communication lines. Once routing was complete, KiCAD's Design Rule Check (DRC) tools were used to verify that all electrical connections were present, no clearances were violated, and all drill and trace specifications were manufacturable.

3.4 Manufacturing

3.4.1 Ordering from JLCPCB

Once the PCB layout was finalized, the manufacturing files were exported from KiCAD in Gerber format. Gerber files are the industry-standard manufacturing outputs used by PCB fabrication facilities to define copper layers, soldermask openings, silkscreen markings, and drill locations. Additional drill and board-outline files were also generated automatically by KiCAD to ensure full compatibility with the fabrication process.

The board was fabricated through JLCPCB, a manufacturing service commonly used by hobbyists, students, and educational robotics teams because of its low pricing and relatively short turnaround times. The order specifications matched the requirements established during the design phase: a two-layer FR-4 PCB with 1 oz copper, a standard 1.6 mm thickness, green soldermask, and white silkscreen labeling.

Because the board measures only 54.5 mm × 29.0 mm, manufacturing costs remained extremely low even when ordering multiple copies. This small size also helped reduce material usage and made the board practical for compact projects or enclosed payload systems. Before submission, KiCAD's built-in Design Rule Check (DRC) system was used to verify compliance with JLCPCB's standard manufacturing tolerances, including the 0.127 mm minimum trace width and clearance rules used throughout the design.

When the fabricated boards arrived, the traces, plated through-holes, silkscreen text, and soldermask alignment all matched the intended layout accurately. The grouped connector arrangement was especially successful physically, as the spacing between ports was sufficient for cables to be inserted and removed cleanly without interfering with neighboring connectors. The final fabricated board therefore validated not only the electrical design, but also the physical usability improvements that motivated the project in the first place.

3.4.2 Soldering Components onto the Board

With the fabricated PCBs complete, the next stage was assembly. The board combines both through-hole and surface-mount components. The connector headers use through-hole pads because they experience repeated physical stress from plugging and unplugging jumper wires, while the capacitors use compact surface-mount packages to reduce board size and simplify routing.

Assembly began with the surface-mount capacitors. The five 0.1 μF bypass capacitors, implemented in 0603 packages, were soldered first because they are the smallest and lowest-profile components on the board. These capacitors were positioned near the power distribution traces feeding the connector groups in order to reduce high-frequency electrical

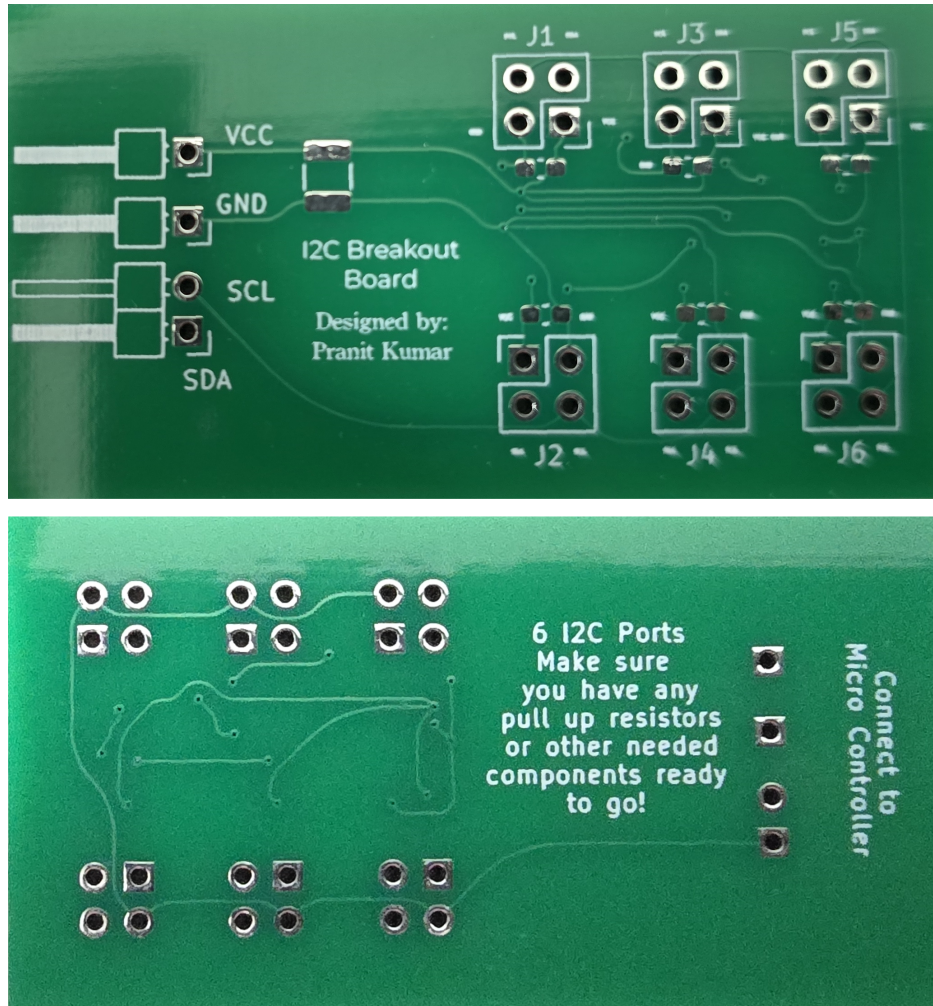


Figure 3: A Printed PCB Model of the Proposal

noise and stabilize local voltage conditions. Afterward, the larger $10\ \mu\text{F}$ bulk capacitor, implemented in a 1210 package, was soldered onto the board near the primary power input region.

Once all surface-mount components were installed, the through-hole connector headers were soldered. The six grouped target-device ports labeled J1 through J6 each use a 2×2 pin arrangement corresponding to VCC, GND, SDA, and SCL. Additional through-hole pads on the left side of the board provide the controller-side input connections for the same four lines. Because these connectors are repeatedly handled during normal use, careful soldering was required to ensure strong mechanical joints capable of withstanding repeated insertion and removal cycles.

Each solder joint was inspected visually after assembly to verify proper wetting and to ensure that no solder bridges existed between adjacent pads. After soldering was complete, continuity testing was performed using a multimeter to confirm that the SDA, SCL, VCC, and GND lines were correctly routed across all six connector groups. Particular attention was paid to ensuring that no accidental shorts existed between the communication lines and

the power rails.

The completed board successfully demonstrated the practical advantages of the grouped-pin layout. Unlike traditional breadboard-based I²C wiring arrangements, cables could be connected, removed, and rerouted from multiple directions without forcing neighboring wires aside. Even when multiple target devices were connected simultaneously, the board remained physically accessible and significantly less cluttered than the traditional row-based approach.

4 Experimentation & Conclusion

4.1 Goal

4.2 Methodology

4.3 Data

4.4 Verdict