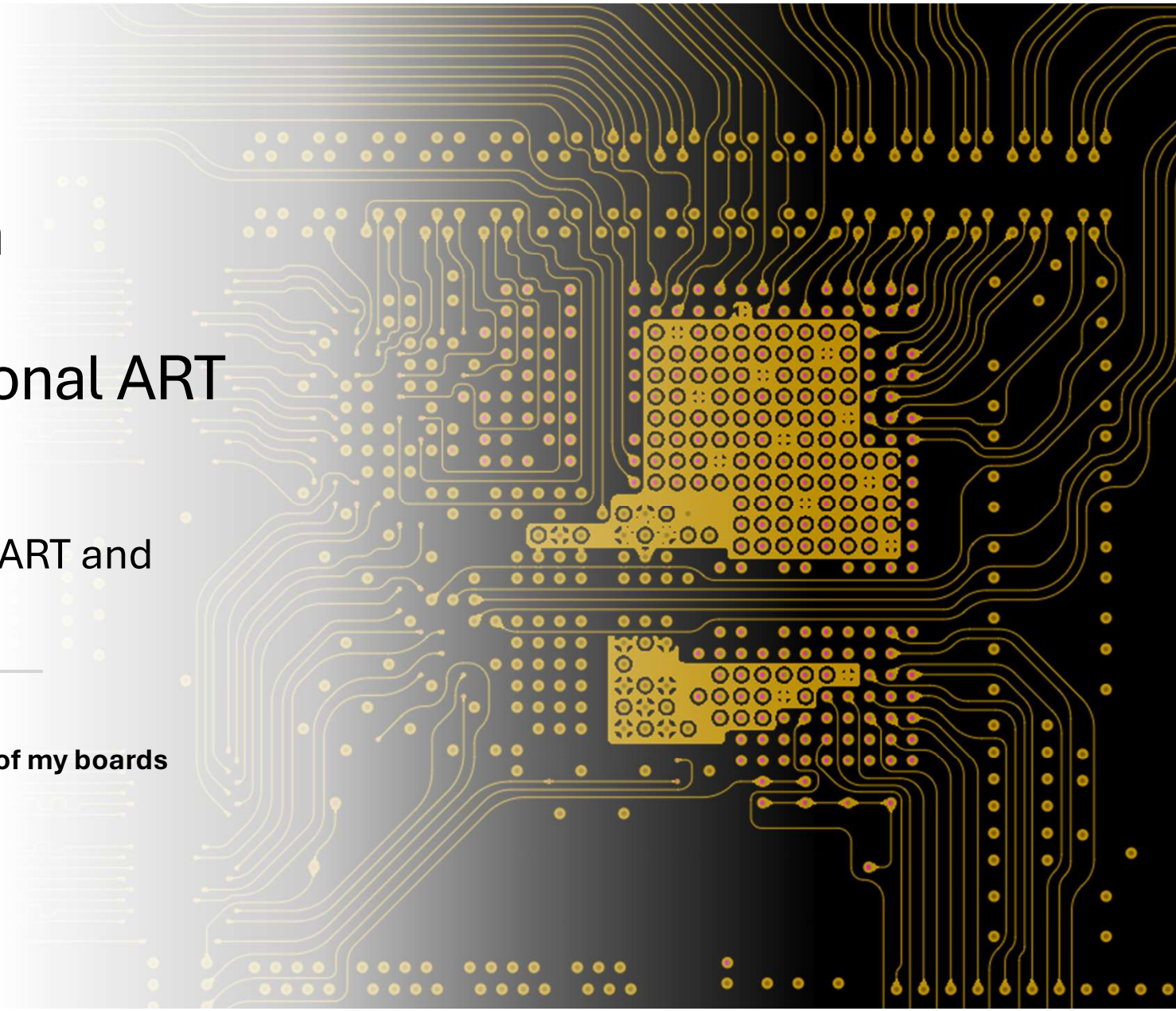
PCB Design

Functional or Functional ART

For me, its about functional ART and here’s WHY

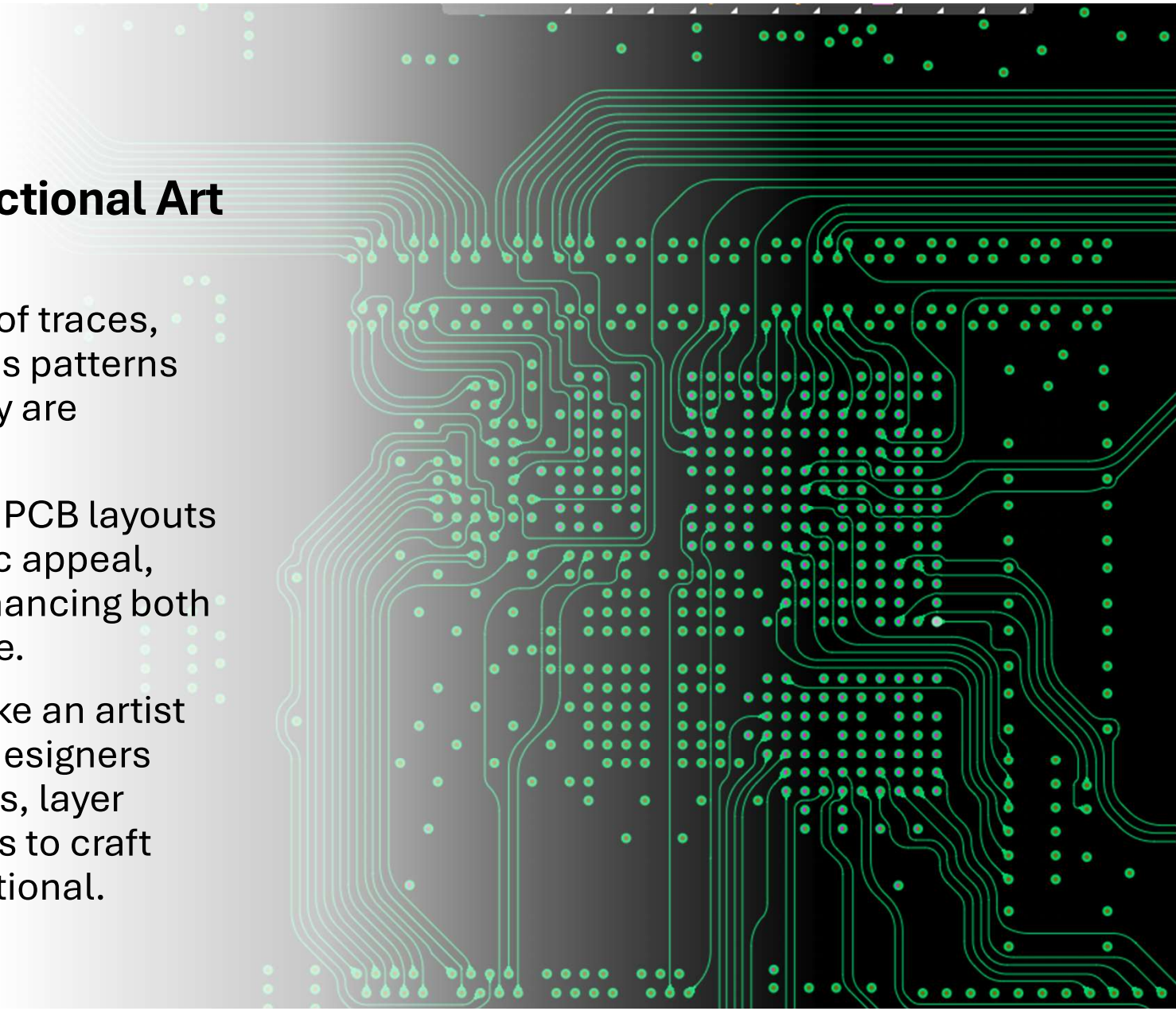
Layout examples in slides are from one of my boards enjoy the organic style routing ☺

Power Point Presentation By
Andy Carpenter 2025



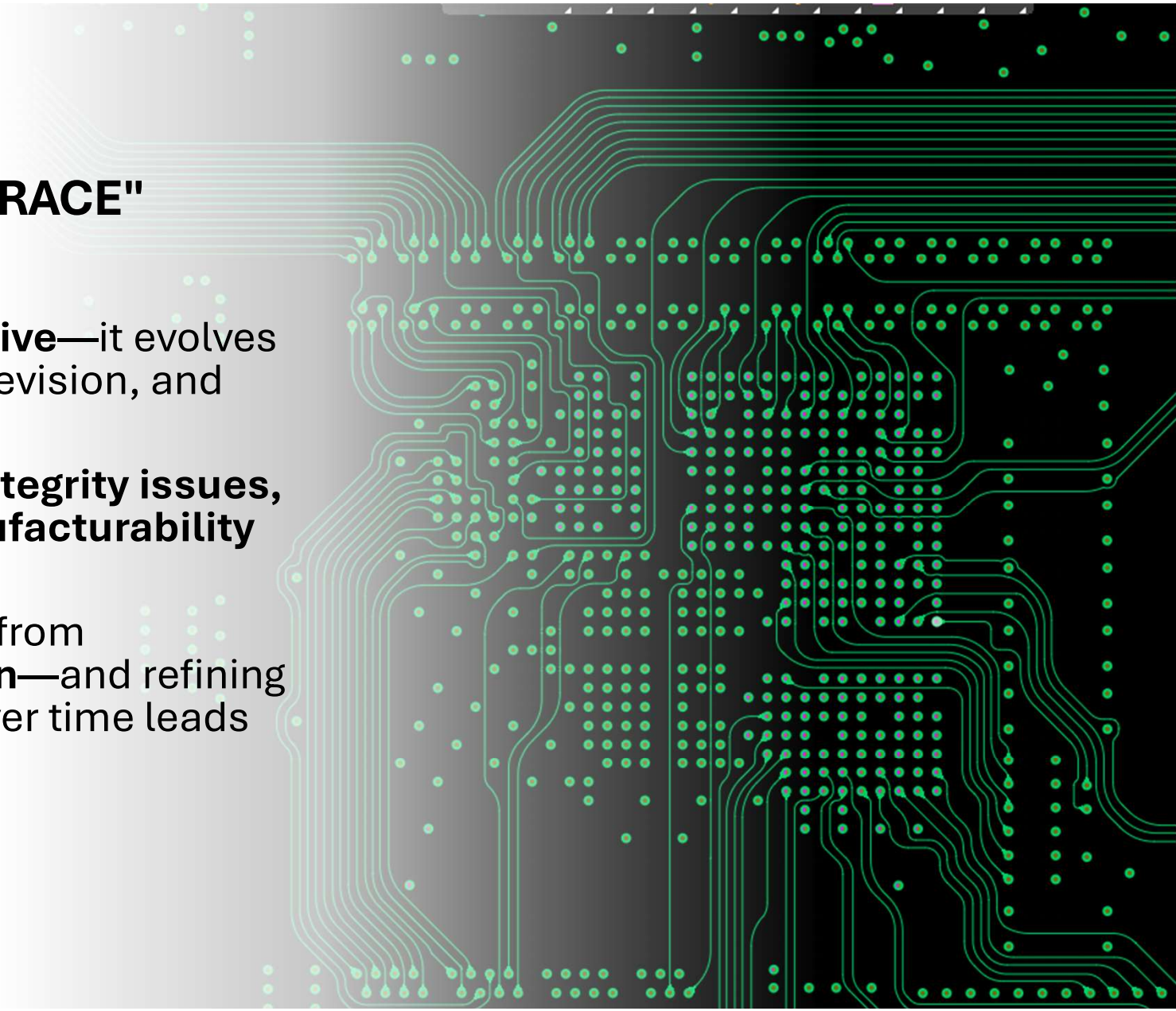
Why PCB Design is Functional Art

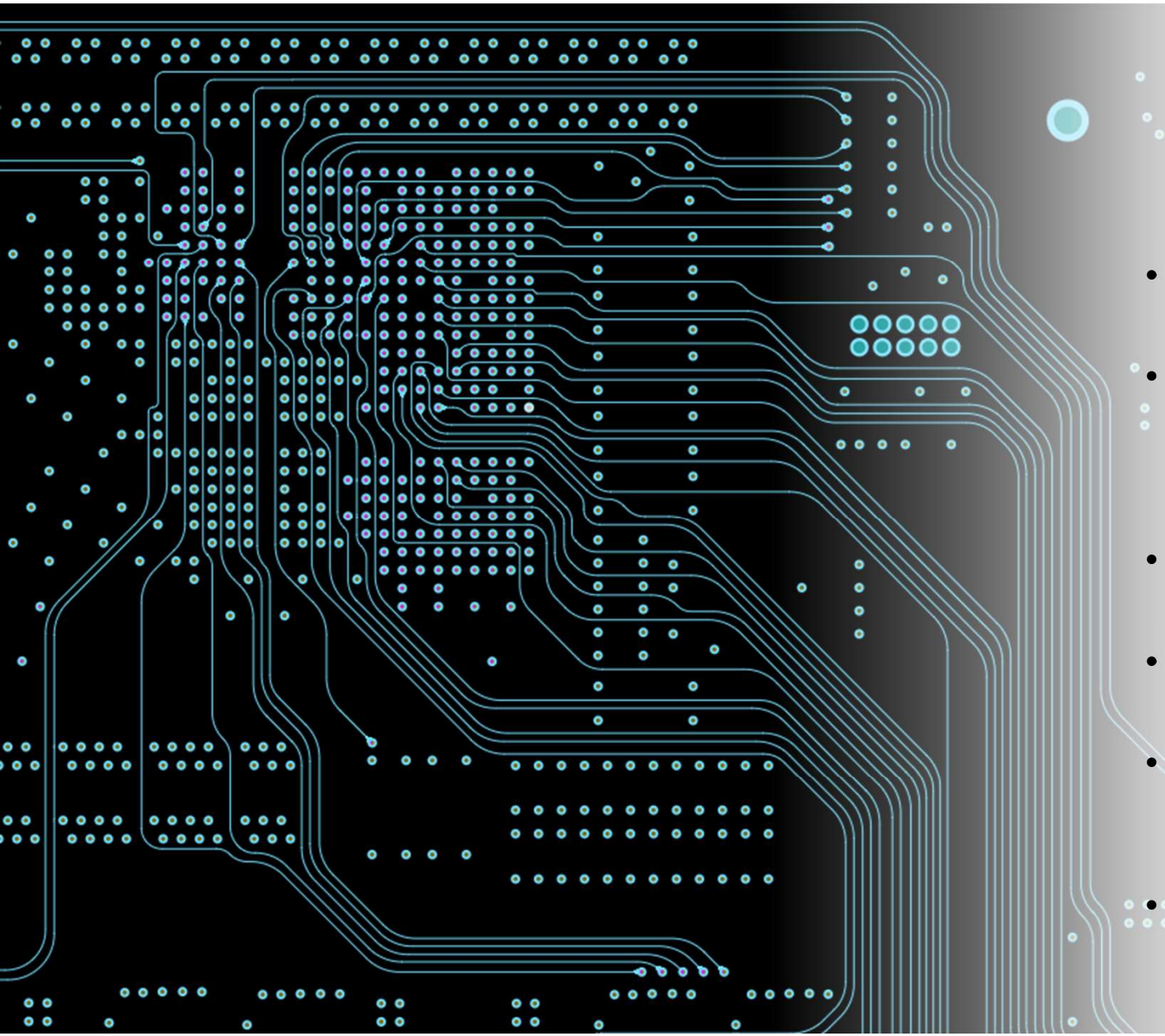
- **Visual Flow**: The placement of traces, vias, and components creates patterns that can be as elegant as they are efficient.
- **Symmetry & Balance**: Good PCB layouts often have a natural aesthetic appeal, with symmetry and order enhancing both performance and appearance.
- **Creative Constraints**: Just like an artist works within a canvas, PCB designers work within board dimensions, layer limits, and routing restrictions to craft something beautiful yet functional.



"ART is a Journey, not a RACE"

- Great PCB design is **iterative**—it evolves through careful thought, revision, and experience.
- Rushing leads to **signal integrity issues, EMI problems, and manufacturability headaches**.
- Each board tells a story—from **conception to production**—and refining your design philosophy over time leads to **mastery**.

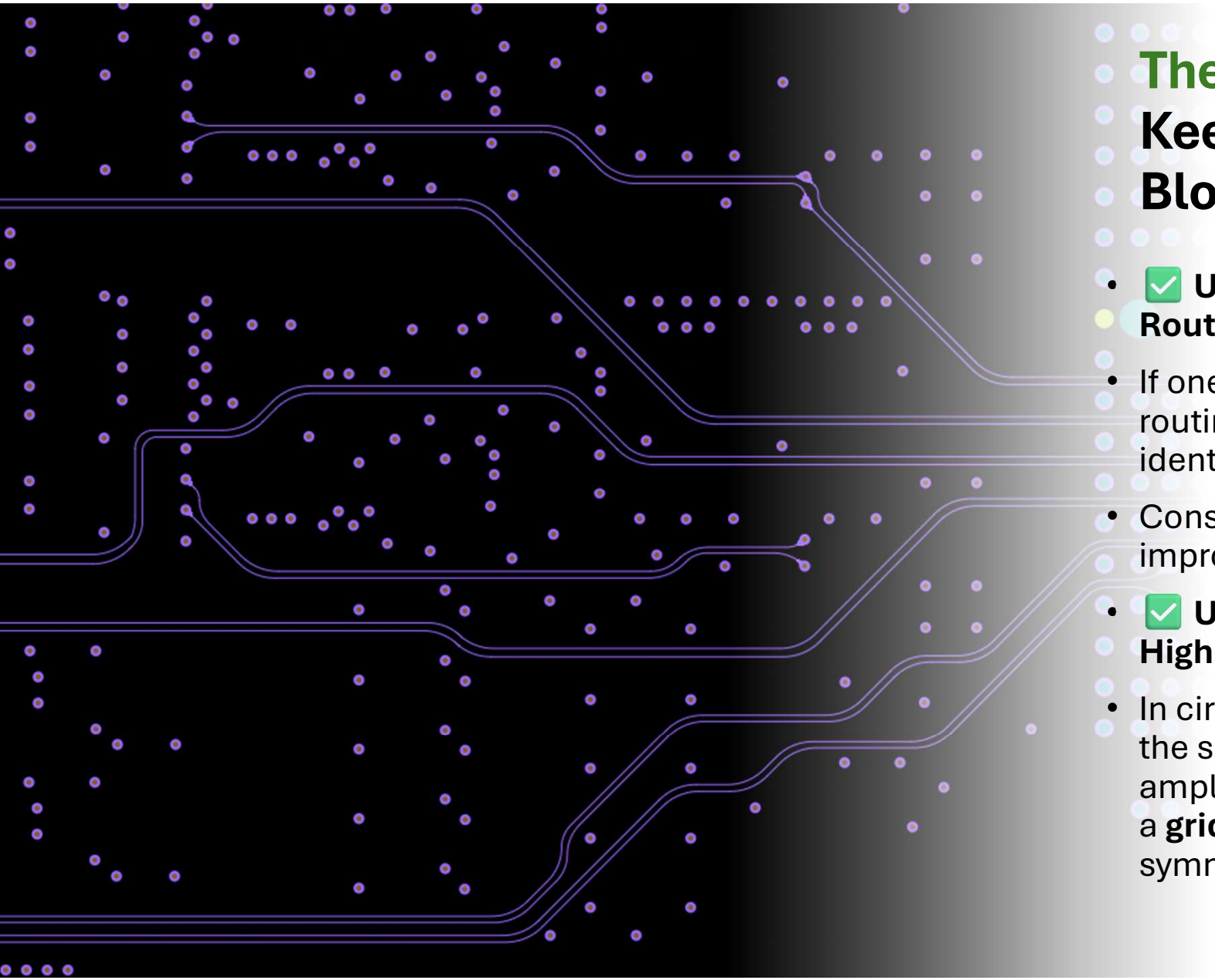




The DO’s

Keep Repeated Circuit Blocks Identical

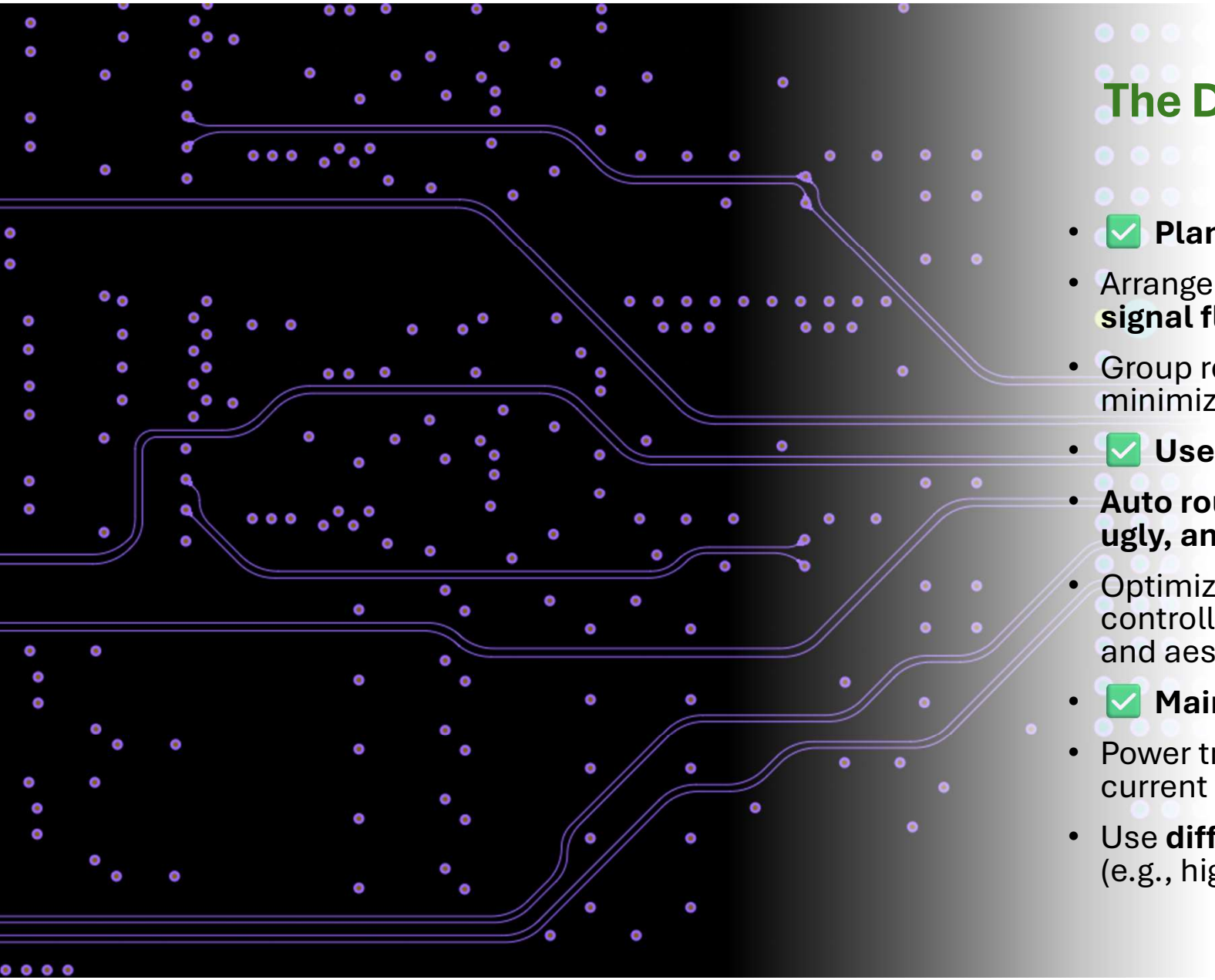
- ✅ **Standardize Layouts for Repeated Functions**
- If a circuit function (e.g., an op-amp filter, power regulation stage) appears multiple times, **copy-paste the layout** to ensure consistency.
- This improves symmetry, simplifies routing, and makes debugging easier.
- ✅ **Align Components and Traces Neatly**
- Keep the orientation and placement of resistors, capacitors, and ICs the same for each instance.
- This **reduces layout errors** and makes the board easier to assemble.



The DO’s

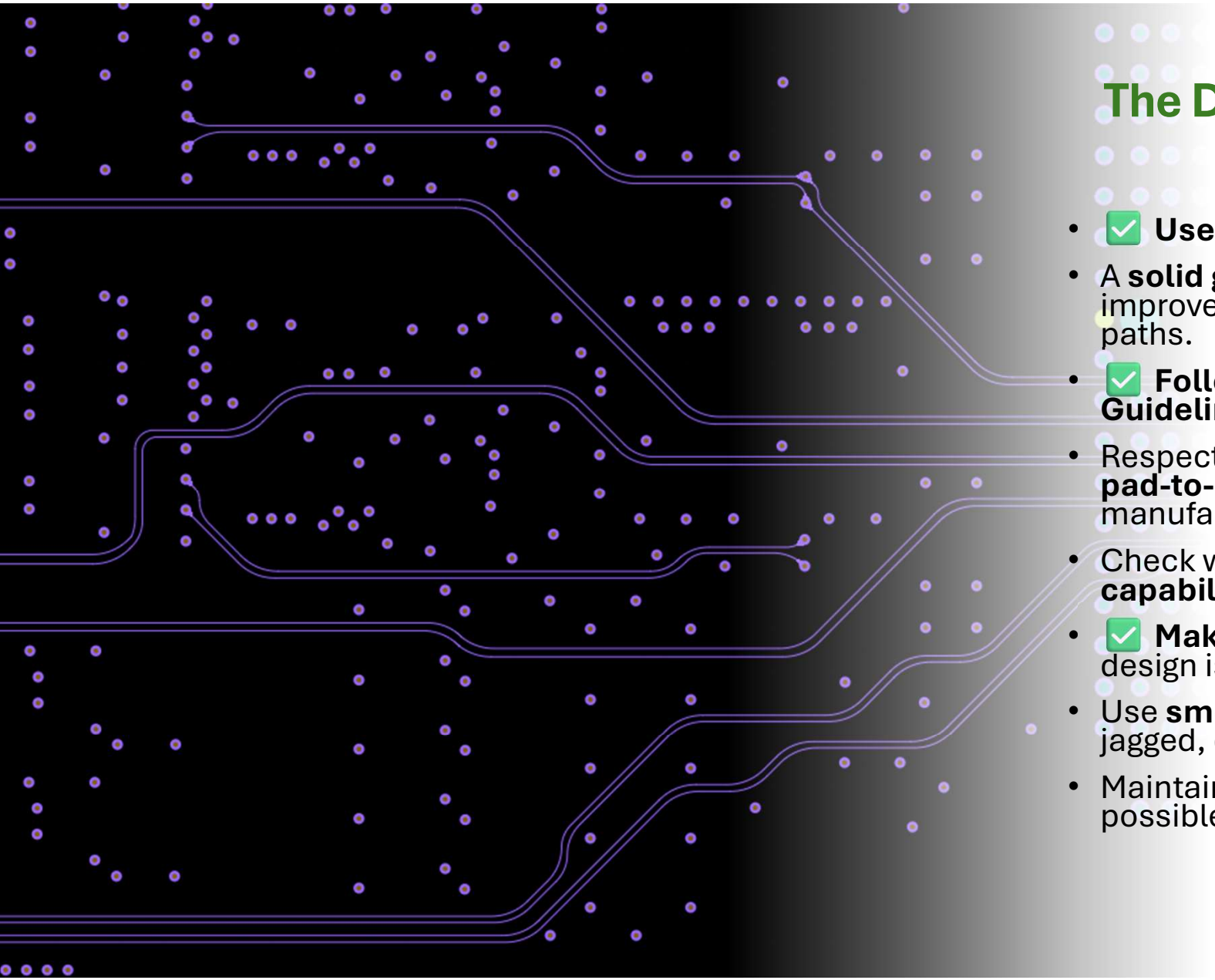
Keep Repeated Circuit Blocks Identical

- ✅ **Use the Same Trace Widths and Routing Styles**
- If one section has clean, optimized routing, **apply the same strategy** to identical sections.
- Consistency reduces parasitics and improves circuit performance.
- ✅ **Use Array-Based Placement for High-Density Designs**
- In circuits with multiple instances of the same function (e.g., LED drivers, amplifier channels), arrange them in a **grid or mirrored pattern** for symmetry.



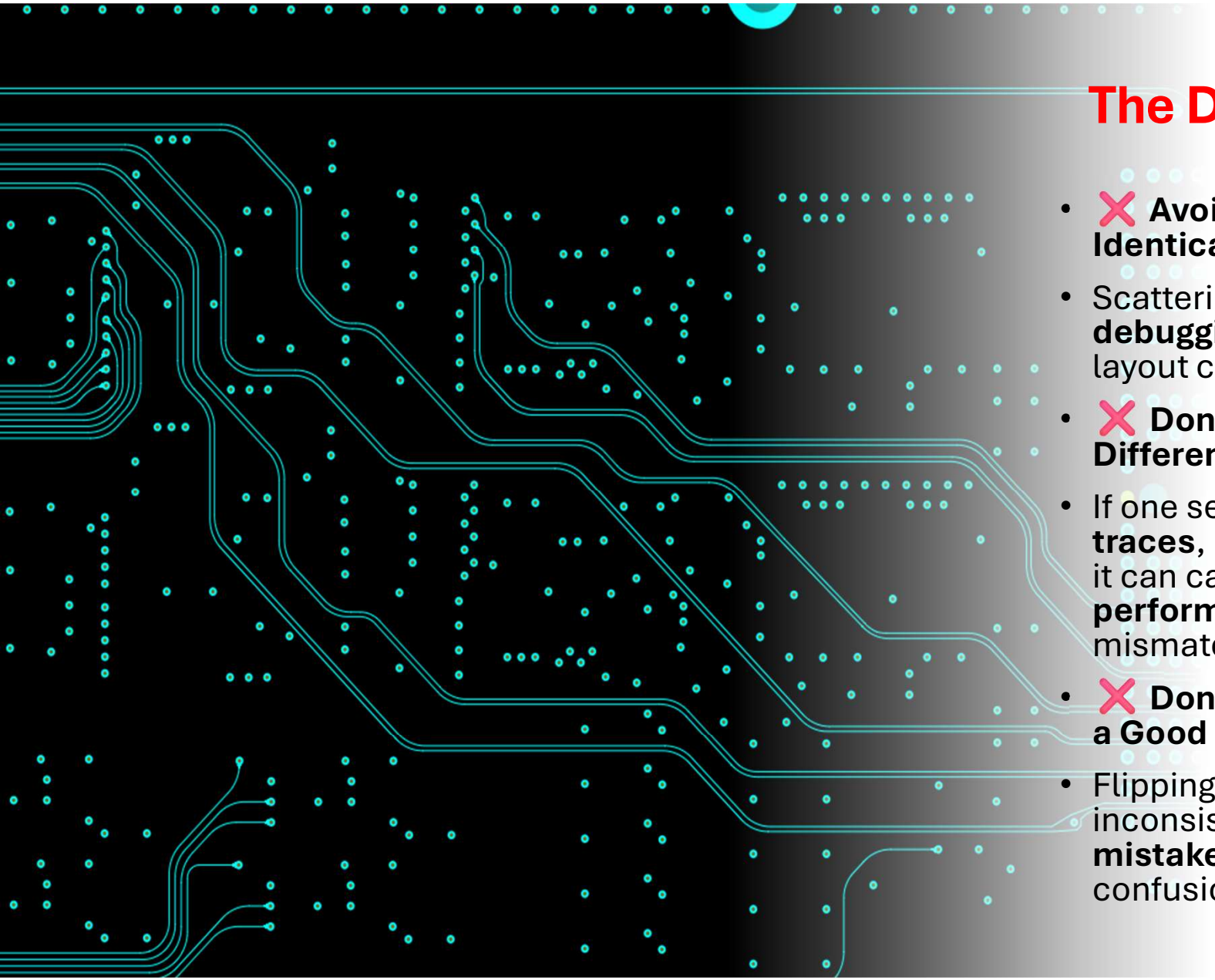
The DO’s

- ✅ **Plan Your Layout Carefully**
- Arrange components logically for **clear signal flow** and **short trace lengths**.
- Group related components together to minimize noise and interference.
- ✅ **Use Manual Routing**
- **Auto routing often leads to inefficient, ugly, and problematic layouts**.
- Optimize trace paths **manually** for controlled impedance, minimal crosstalk, and aesthetics.
- ✅ **Maintain Consistent Trace Widths**
- Power traces should be **wider** to handle current without overheating.
- Use **differential pairs** where necessary (e.g., high-speed signals).



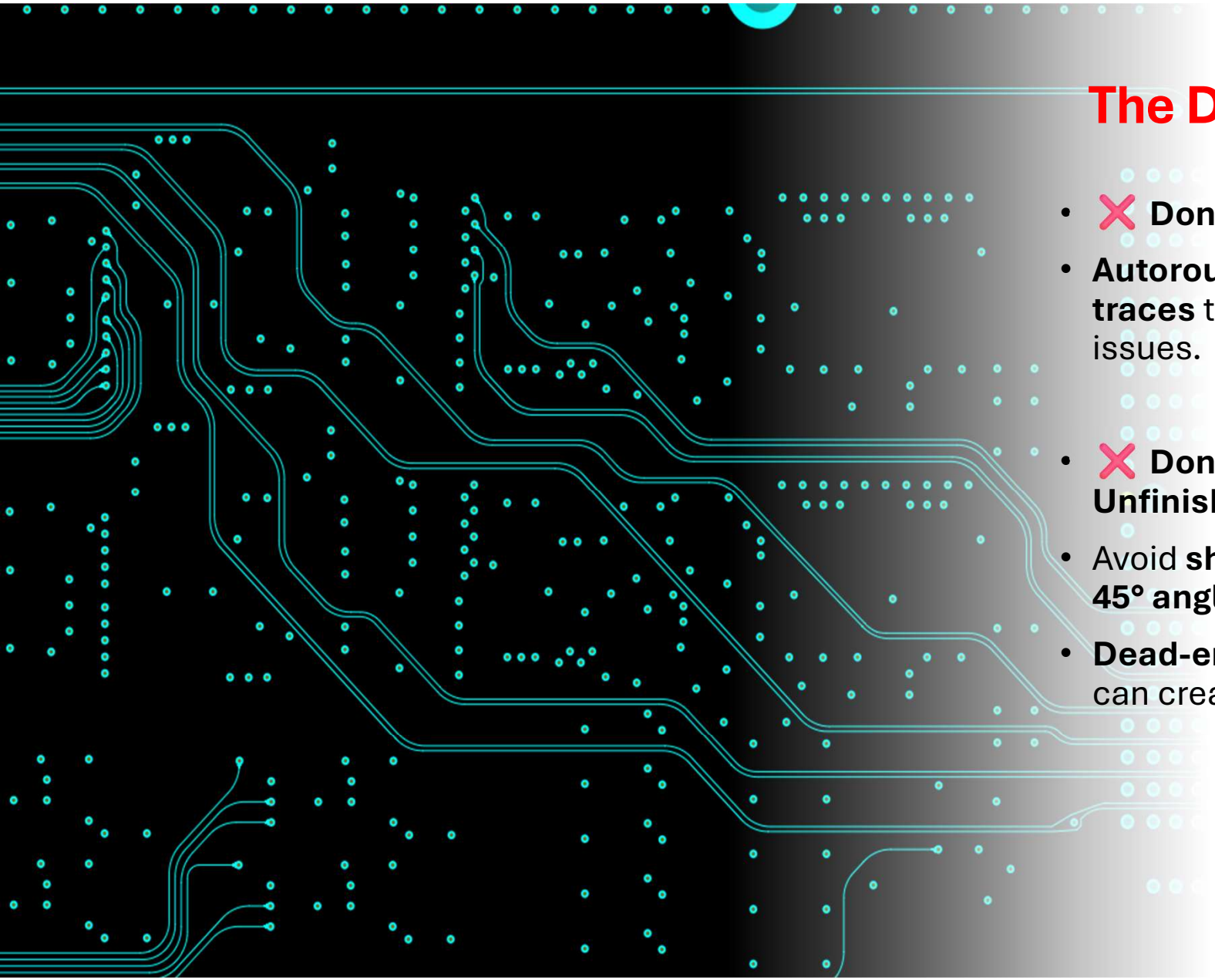
The DO’s

- ✅ **Use Ground Planes**
- A **solid ground plane** reduces noise, improves EMC, and simplifies return paths.
- ✅ **Follow Design Rules & DFM Guidelines**
- Respect **trace clearance, via sizes, and pad-to-pad spacing** to avoid manufacturing issues.
- Check with your **PCB manufacturer’s capabilities** before finalizing.
- ✅ **Make it Look Good** (Because PCB design is ART!)
- Use **smooth, flowing traces** rather than jagged, disorganized paths.
- Maintain **symmetry and balance** where possible for a professional look.



The DON’T’s

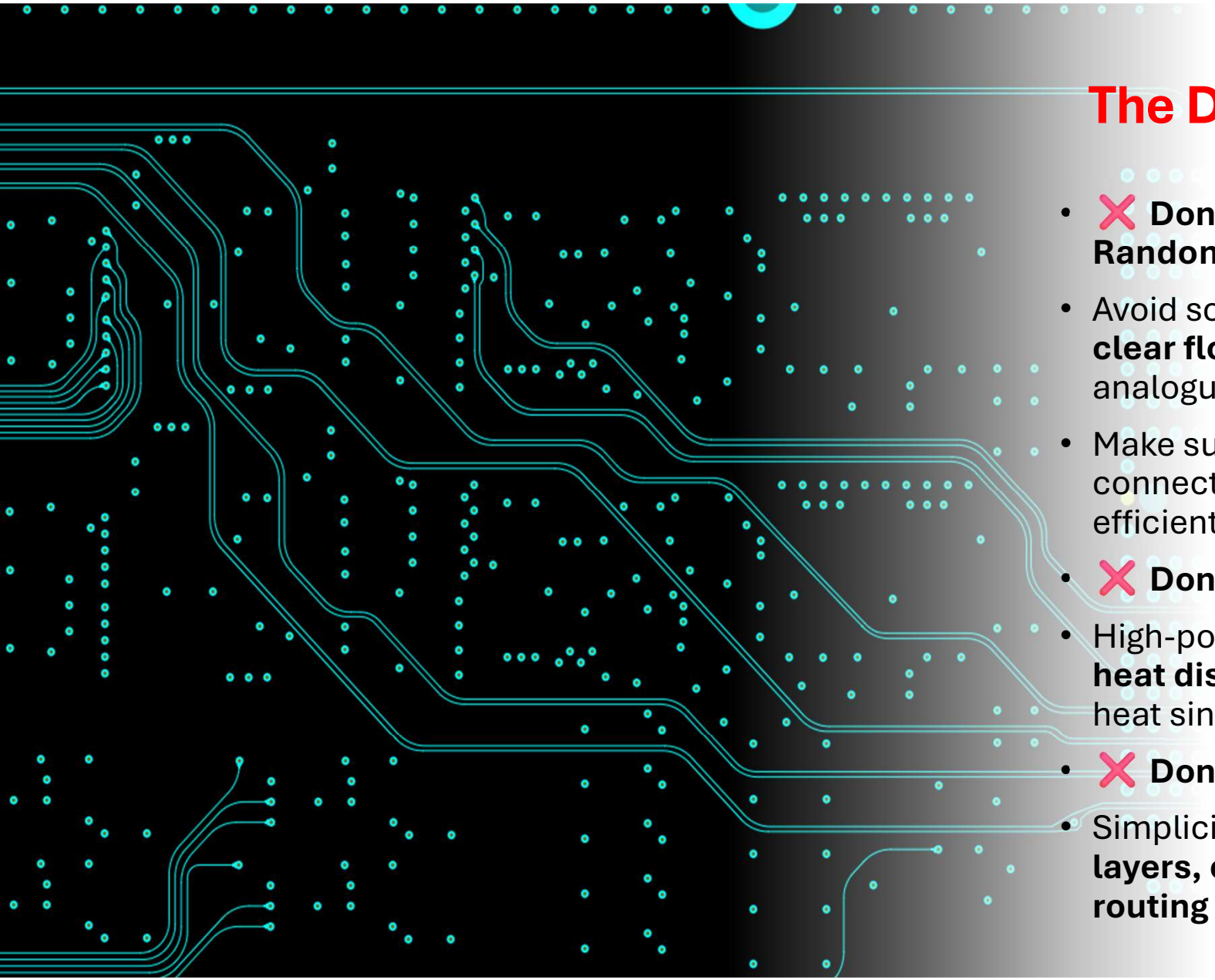
- ❌ **Avoid Random Placement for Identical Circuits**
- Scattering repeated blocks **makes debugging difficult** and increases layout complexity.
- ❌ **Don’t Route Identical Blocks Differently**
- If one section is routed with **clean traces**, but another has a messy layout, it can cause **uneven electrical performance** (e.g., signal delay, mismatched impedances).
- ❌ **Don’t Change Orientation Without a Good Reason**
- Flipping or rotating similar sections inconsistently can lead to **assembly mistakes** and troubleshooting confusion.



The DON’T’s

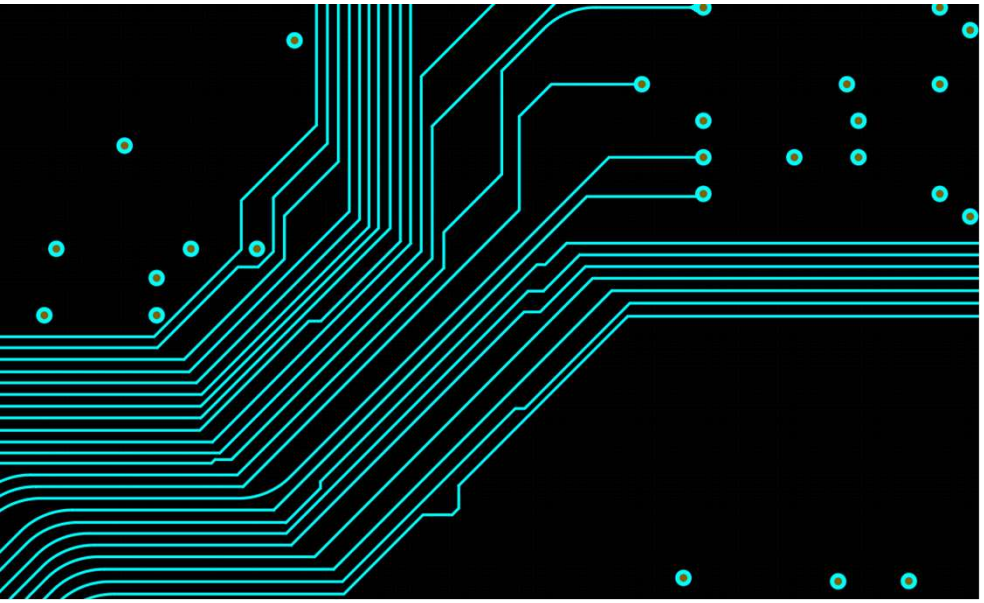
- ❌ **Don’t Autoroute Everything**
- **Autorouting creates inefficient, messy traces** that can increase signal integrity issues.

- ❌ **Don’t Leave Traces Messy & Unfinished**
- Avoid **sharp angles (90° turns)**—use **45° angles** or arcs to reduce EMI.
- **Dead-end traces** or poorly routed paths can create signal integrity problems.



The DON’T’s

- ❌ **Don’t Place Components Randomly**
- Avoid scattering components **without a clear flow**—keep high-speed and analogue sections separate.
- Make sure power and ground connections are easy to route efficiently.
- ❌ **Don’t Ignore Thermal Management**
- High-power components **need proper heat dissipation**—use thermal vias or heat sinks.
- ❌ **Don’t Overcomplicate the Design**
- Simplicity is key: **avoid unnecessary layers, excessive vias, and complex routing** that makes debugging harder.



The Designer just CAREs about FUNCTION

JUST FUNCTION MEETING THE MILESTONE

The above is not a real PCB but it is typical of what I sometimes see. I see engineers designing a functional board without CARE or CONSIDERATION

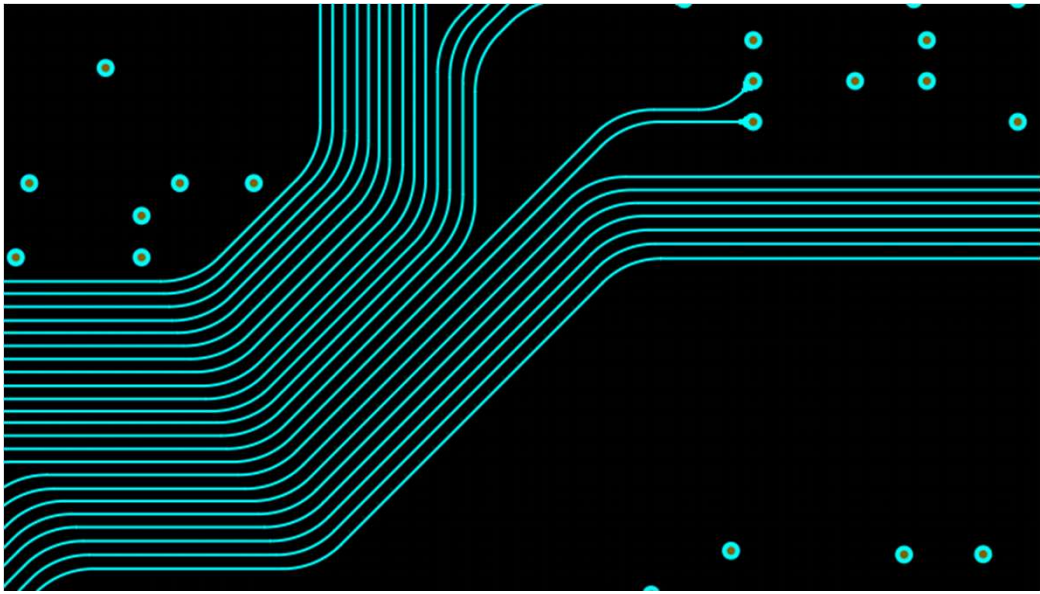
- Inconsistent spacing
- not a very smooth flowing routes

FUNCTIONAL ART EXAMPLE

The below is not a real PCB but it is typical of what I would like to see. Engineers designing a functional board but with CARE and CONSIDERATION

- consistent spacing
- smooth flowing routes

Designer CAREs about his WORK



Conclusion

- A lot of engineering tasks are not tangible they are not seen or can’t be held. PCB design is uniquely different.

- It can be seen
- It can be held
- It must function / perform

So, design your PCB’s so they function but also so you can hang them on your wall like a Picaso or a Banksy!!!