



Need a quote for a project? Call 720-778-2400

Get Quote!

Growing Season

With back-to-school season and sweater weather upon us, we're turning over a new leaf this Fall. The past 6 months have been marked by rapid growth, and we're taking stock of what we've been able to accomplish and improve upon recently in order to make the PCB prototype and production process more seamless and less time-consuming for our customers.

So what's new?

First, we've revamped our production planning process thanks to advanced software that dramatically improves the way we plan and move jobs through the floor.

Second, we added a power conditioner to improve power reliability at our Denver assembly facility. This has significantly improved equipment up-time, ensured less delays due to power outage, and resulted in the vast majority of jobs being completed on or ahead of schedule.

Finally, rapid growth goes hand-in-hand with staffing changes, and we're proud of how our team has grown over the past few months. We added a second shift to our production team, so we can take on more jobs while ensuring the same quality, attention to detail, *and* quick-turnaround times that our customers know and love. And, we've implemented a plan for continuous improvement company-wide, a change that empowers team members to identify areas of improvement and make necessary changes in a way that is collaborative, effective, and ongoing.

In addition to improved capacity thanks to all of the above, we recently earned AS9100 certification at our Wisconsin facility, a hard-earned achievement that allows us to better serve the Aerospace and Defense industries.

New technological advancement is on the horizon, too, as we will soon be operating a brand new selective solder machine on our Denver assembly floor.

It's an exciting time for the SlingShot family, and with a new season just starting, we want to thank our customers and our partners who have been with us from the start. We are grateful to enter this new and exciting chapter with you!

Engineering and Design Tips & Tricks:

1. Including CAD data with your Gerber files increases DFA accuracy and speed
2. Call out wash sensitive parts on your assembly drawing or in the quote request
3. Include alternate engineering contacts if you will be unavailable during the assembly time.

Forget Gerbers; Intelligent CAD Files are Better for PCB Assembly

The Gerber file format was first defined in 1980 for vector photoplotters. As the use of computer aided manufacturing (CAM) for printed circuit board (PCB) fabrication grew during the eighties, the Gerber file format was adopted by most manufacturers as the de facto standard for CAD-to-CAM data exchange. In the nearly 40 years since it was first defined, the Gerber file format has evolved and improved some, but newer formats exist that are far superior. It's time for the industry to take a leap forward and leave Gerbers behind.

Read more [on the blog](#).



sales@sassembly.com
www.slingshotassembly.com



+1-720-778-2400



6776 E. 47th Ave. Dr.
Dever, CO

Get Quote!



