

That etch resist technique doesn't work too well, does it?

The "newer processes" are actually no different than the older processes when it comes to fiberglass PCB manufacturing. They have offered various products for years, from single-sided and double-sided to multiple layers, with or without a solder mask and/or silkscreen. The solder mask is the green coating that prevents the solder from sticking where it isn't wanted, mostly done for wave soldering but usually there on high-quality boards even if volume is low (and therefore will be hand-assembled). The silkscreen is there to help identify component locations. It's usually done in white, mostly part identification numbers and sometimes outlines of where some parts go.

What is newer is the CAD software used to deliver the artwork to the board house. It used to be done photographically, and your layout engineer would "lay tape". Literally. They used strips of colored translucent tape fastened to large clear sheets of plastic, one per layer. They often used different colored tape so when each layer was laid on top of each other, you could see the layers easily by color coding. But this was just for convenience while developing and checking the layout, because the final product of the layout engineer is a photographic positive done in black on clear at scale for each layer requested.

Now days, this is usually done electronically by sending a CAD model of the layout in a standard Gerber format. You can design the board using OrCAD or something similar, and then send the Gerber file to the board house for quotations. They usually have an inexpensive prototype trial run these days (which they didn't have in decades past), so you ask for a relatively small number. If you approve the quote, they'll use the Gerber file to make the boards and send 'em to you. The inexpensive trial runs are nice, because back in the day, you didn't have that luxury so you better get it right first time. Otherwise, you had "Engineering Change Orders" (ECOs) that essentially meant you cut a trace and if it needs to be rerouted, you soldered a small wire-wrap wire onto the board as a replacement conductor. It could be a bear to find a problem in a first-run too, especially in complex designs.