Subject: Re: Point to Point vs PCB
Posted by Wayne Parham on Sat, 17 Apr 2021 15:04:17 GMT
View Forum Message <> Reply to Message

The benefits of the solder mask, silkscreen and plate-through holes are not qualitative, not at least as far as the electrical signals go or even in terms of mechanical strength or anything like that. A single sided, one-layer board without solder mask or silkscreen is fine in many cases.

The solder mask is there for one purpose: To prevent solder from adhering to places where it isn't wanted, and its primary benefit is when using automated soldering systems like wave machines.

Silkscreens are just to add labeling.

Multiple layers allow the use of more complex boards without using jumpers. It allows signal traces to cross one another by going over another land on the opposite layer. There are even multi-layer boards that have several layers laminated together for extremely complex boards. The connection between layers is done with plate-through holes.

There are things that none of these features address that are more important qualitatively. All involve layout. The thickness of the lands sets the current capacity. Inter-conductor impedance and signal coupling are determined by proximity and orientation of the lands with respect to one another. So noise can be reduced or even eliminated by proper layout, and the converse is also true. A poor layout can create a noisy system design. Electrical noise presents itself as hum or buzz, or as artifacts in a video signal. While this is annoying in an audio or video circuit, it can cause a digital circuit to malfunction, sometimes intermittently where it is hard to track down.