NO TOUCH PCBs

Design Guidelines/Specs

Summary	2 layer, 4 layer and 6 layer ROHS-compliant standard technology PCBS instantly Quoted and orderable online with automatic design file verification.
Data Required	Gerber files in RS-274X format only (Extended Gerber) or ODB++ format. Do NOT use Gerber X2 format! AND Excellon drill file in ASCII format with the following parameters: • Units = Inch • Tool Units = Inch • Coordinate = Absolute • Zeros omitted = Trailing • Number format = 2.4
Turn-time	1, 2, 3, 4 and 10 business days.
Cut-off Time	5 PM PST (all orders coming in on a given weekday until 5 PM will start next weekday as Day 1).
Minimum Order	No minimum quantity. However, the more you order, the more you save.
Maximum Area of a Board	100 sq" Note: The board X or Y dimension should not exceed 15.9" or be less than 0.5".
Material	FR4 Lead-free only.
Thickness	0.062" (+/- 0.006). No special constructions allowed. We will use our standard constructions that lead to correct overall thickness.
Surface Finish	HASL or ENIG (RoHS-compliant)
Minimum Hole Size	0.008" (+/- 0.003) finished hole size.
Minimum Trace & Space	0. 004"
Minimum Width of Annular Ring	0.005" (meaning, your pad diameter must be at least 0.01" larger than the hole diameter).

Standard Nominal Copper	1 oz/sq foot on outer layers 0.5 oz on inner layers
Maximum Hole Density	50 holes per square inch (calculated as: total holes on individual board Divided By area of one board)
Maximum Plated Hole Size	0. 246"
Maximum Non-Plated Hole Size	0.250 inches (250 mils). Also note that non-plated holes must not touch any copper.
Soldermask	Green LPI; openings in the solder-mask layers should be 0.006" larger than the copper feature.
Legend/Silkscreen	White color; both Top and Bottom allowed. (Please ensure the legend lettering width is minimum 0.005")
Restrictions	The following are not allowed in No Touch PCBs: • Arrays • Internal cutouts • Steps and Repeats • Milling • Blind/ Buried vias • Multiple Parts or Part numbers ($10 charge applies to each additional part) **Outline and Corners** Board outline is required for all orders and should be placed on one of your "working" layers. It must not have more than 16 right angles. The board shape must have at least 1 straight edge, which should be at least 0.25" long. For rounded corners the radius of the corner must exceed 0.05". **Filled Areas** Crosshatching is not allowed; many CAD systems leave gaps in fill areas when using 45 degree lines. For No Touch, you must use 0 degree and/or 90 degree lines to do the fill. Please make the filled areas using 0.01" outlines with 0.001"overlap. **Notches** Notches are allowed: you can have up to 6 notches, but every notch must be at least 0.1" wide. If a notch with is less than 0.1", our router may ignore your spec and make it our minimum width.
Electrical Testing	Not offered. See our **Web PCB Service** product (www.protoexpress.com/webpcb).
Shipping options	FedEx, UPS, Will Call. You can use your own FedEx Account number. International Shipments allowed. All duties/customs are responsibility of the customer.
Payment Terms	Credit Card only.

Data Guidelines	Customer must be careful about accurately doing layer assignment in the online system. Customer must also accurately put in polarity (positive or negative) information for the layers. See FAQ's online for details on how we define positive vs negative polarity for different layers.
Additional Requirements	All layers must be aligned and not mirrored. The layers should provide a view as if you were looking at it from top straight down to the bottom. To ensure this, we recommend double-checking this through a Gerber viewer and not just rely on the view from your design software. Occasionally, we notice the misalignment problem before the board is built, and we may correct it for you. But, generally, it is **YOUR** responsibility to make sure your layers are aligned correctly and not mirrored. Note: We put a small 6-digit order # on each board for easy identification during fab process. This will either appear on the top or bottom silkscreen or etched into the bottom copper layer.
How to Quote and Order	1. On www.protoexpress.com website go to No Touch Service, under Products & Services menu 2. Upload your Gerber files 3. Assign Layer and Drill information (easy to do) 4. Wait a few minutes for our Auto File Verification (AFV) program to check your design. 5. Place an order online using credit card.

www.protoexpress.com/notouch