"Why change from EXPRESS*** design tools to EAGLE PRO?"

While the ExpressSCH and ExpressPCB tools are free, the primary reason why so many engineers prefer them to the expensive high-end design tools, there is a significant drawback from taking this path. The most significant issue with using Express design tools is that the resultant Printed Circuit Boards (PCB's) must be made by only the company supplying the tools, ExpressPCB.

There are dozens of PCB manufacturers, hundreds worldwide, that can create competitive solutions if they are provided the GERBER and EXCELLON files needed to create PCB's. These files are NOT available as export-output-files from the ExpressPCB tools, preventing competitive purchasing of these boards.

Almost all of my clients want these design files so that they can purchase PCB's from any fabricator they choose (*often as part of a full turn-key package at a Contract Manufacturer*). Many of my clients want designs that require 'bleeding' edge approaches, including 6 and more layers, very tight specifications on trace width and separation (*less than 6mils*), small (*sometimes laser*) drills (*less than 14mils*), non-green soldermasks, non-white silkscreens, and sometimes blind and buried vias for signal obfuscation, special surface treatments, thick-out-copper layers for high-current applications, and other features that are not possible at the ExpressPCB board house.

Using the EAGLE PRO tools enables the design of PCB's without restrictions, within Design Rules per board fabricator, of course. The GERBER and EXCELLON CAM files <u>can be sent to any board fabricator for quotation and manufacture</u>. There are many other EAGLE-based design support output files that enable easier BOM building, robot/machine assembly of populated boards, statistics and information files, etc., all to reduce the time and increase the accuracy and efficiency of creating the final product, a properly assembled board.

Free downloadable versions of the EAGLE tools also enable the clients to examine the design they have purchased (*from GUSTECH*), and in most cases make changes themselves; taking full ownership of their board design as possible.

As a footnote, I would also like to point out that I still have a couple of clients that prefer to have their designs created using the ExpressPCB tools. Therefore, I am also still using current (*version 7.4.1*) ExpressSCH and ExpressPCB tools for those who still might be interested in this approach.