

Proteus Version 8.7

- ✓ High Speed Design
- ✓ Placement Rooms

- ✓ Automatic Via Stitching
- ✓ Library Management

Introduction

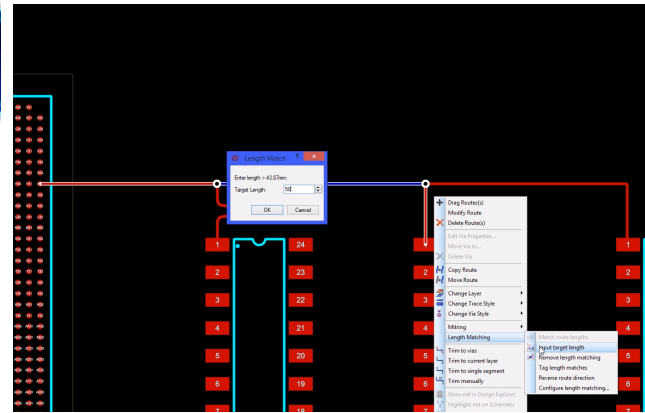
We are pleased to announce that Proteus V8.7 is now nearly development complete and we are expecting to release a public Beta shortly. This is a significant release with a focus on simplifying common tasks during PCB Design.

High Speed Design

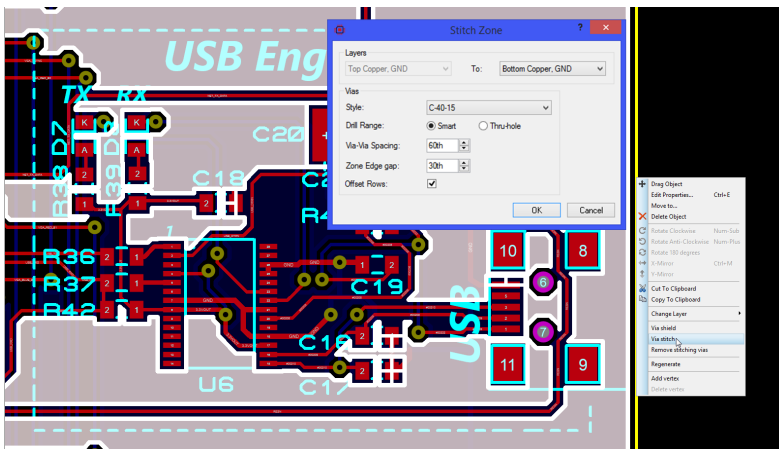
With Proteus 8.7 we have continued to improve our support for high speed design, building on the automatic length matching work we started in V8.6. This includes support for complex length matching where one section of track is part of more than one length matched route (as commonly seen with fly-by routing topologies).

We've also included the ability to add internal lengths to footprints on the layout. This enables more accurate matching for critical signals, particularly with BGA's where the signal delay inside the chip can be relevant.

Finally, we've also added navigation and browsing of length matched routes from connectivity highlight mode and reworked the length match report file to contain more information. [Watch the video demonstration here.](#)



Automatic Via Stitching and Shielding



Via stitching is a technique used to tie together larger copper areas on different layers, helping keep return paths short and to reduce noise on the PCB. You can automatically stitch planes in Proteus via a simple context menu command. This gives you control over via style and also spacing and row offsets for the stitching pattern.

Once placed you can move and edit vias in the normal way. You can remove groups of vias with a selection box and the selection filter or you can remove all stitching vias with the remove stitching command on the zone context menu.

A via shield or picket fence can be added around the border of the planes in much the same way. This will create a single row of vias around the perimeter of the zones and can help prevent electromagnetic interference with other equipment.

The same technique can be used with high speed routes such as microstrip or stripline to help isolate signals on the PCB operating at different frequencies.

