# PCB Layout

## Single environment for placement & routing of today's most complex PCB designs

**Overview**

Xpedition® xPCB Layout, formerly known as Expedition PCB, combines ease-of-use with advanced functionality, enabling designers to create the most complex of PCB designs.
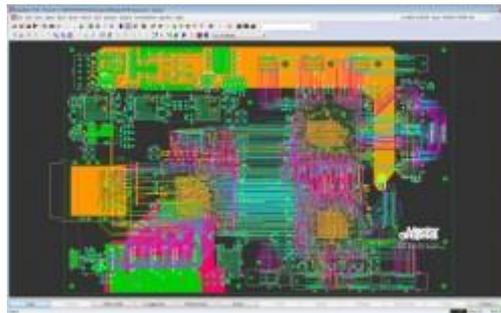
Its interactive and customizable multi-pass autorouting controls include:

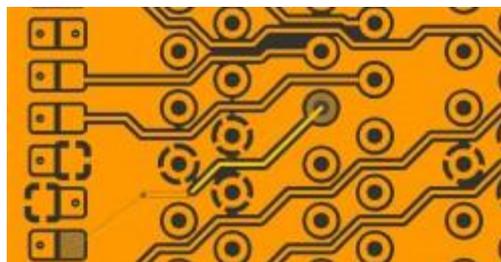Differential pair routing

Net tuning
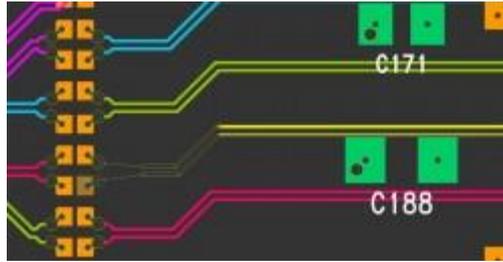
Manufacturing optimization

Microvia technology



**Single, integrated place and route editing environment**

Helps reduce total design time and increase productivity by providing a single environment for critical layout tasks



**Correct-by-construction design methodology**

Helps designers produce high-quality results and eliminate clean-up time

**Addresses your most important design challenges**

Provides all the tools designers need to handle complex design challenges including differential pair routing, net tuning, manufacturing optimization, flex circuits, embedded passives, and microvia technology

**Technical Specifications**

- Common constraint definition environment is shared between schematic capture and layout, allowing the evaluation of critical signals at any design stage
- Autorouting technology combined with interactive editing capabilities produces a single, powerful and easy-to-use design environment
- Common database and user interfaces and rules increase productivity
- The RF module integrates RF and PCB design; the PCB layout environment understands RF circuitry and interfaces directly with Agilent and AWR RF simulation tools
- Clears area fills automatically around traces, vias, and pads as the board is edited
- Rules-by-area greatly improves routing around BGAs and other fine-pitched parts
- Multiplow functionality allows you to simultaneously route multiple nets, including differential pairs, with true 45 degree routing, including routing through areas of staggered pins
- Common constraint definition environment is shared between schematic capture and layout, allowing the evaluation of critical signals at any design stage
- The design reuse module creates and stores reusable blocks of circuitry, including schematic and PCB placement and routing data
- The variant management tool enables the creation of multiple product configurations from a single design database