The Demands of Electronic Product Design

Welcome to the world of board design in Altium's electronic design software. Inside every electronics product is a printed circuit board, or PCB. In the early days of printed circuit board design the demands on the PCB were simple, the board provided mechanical mounting for the components and connected the appropriate pins together - implementing the logical design defined on the schematic.

Today, the components have shrunken so much they are measured in fractions of a millimeter rather than centimeters, and track widths have shrunken from being 10 mil wide, well spaced lines, to thin, 2 or 3 mil hair-like lines tightly packed together. Rising signal speeds has also seen the PCB interconnects change from being simple copper conduits that carry electrical energy, to behaving as high-speed transmission lines, requiring design techniques to cater for this. Lower device supply rails introduces tight demands on the allowable voltage drops, requiring careful design of the power distribution network.

Combine these, and the modern PCB is a dense, multi-layered engineering design challenge.

As well as these more demanding electrical requirements, the mechanical requirements have also become more complex. Compact, unusually shaped modern electronic products require compact, unusually shaped printed circuit boards which are often implemented as rigid-flex structures - these boards can have curved edges and cutouts, requiring carefully positioned components. These tight design requirements demand close collaboration between the electronic design and mechanical design domains, demanding the easy passage of design data between them.

Solving the Design Challenges

These challenges can be met with Altium's PCB design technologies.
Boards can be made in any shape, and designed as a rigid board, rigid-flex or pure flex. Multiple layer stackups can be defined in a rigid-flex design, and additional coverlays created. Components can be embedded on any internal signal layer.

The board can have up to 32 signal and 16 plane layers. If required, plane layers can be split any number of times, and split-within-split areas defined.

32 mechanical design layers are available, which can also be paired and used as specialized fabrication layers, for example, for glue dot definition.

The board shape can also be defined externally, and imported into the PCB editor via DXF, DWG or STEP.

Read about The Board
A Flexible Workspace with Cartesian and Polar Grids

The designer can work with metric or imperial, cartesian or polar grids. Multiple snap grids can be overlaid and can also be restricted to just objects, or just components. The board can range in size from the miniscule, right up to 100x100 inches, using design objects down to 0.001mil in size.

Complimenting the snap grid system is a hotspot snap feature, overriding the snap grid and pulling the cursor to an object's hotspot when it is within the user-definable range. This feature makes it easy to work with off-grid objects, for example routing up to the pads of an imperial component on a metric board.

There are also user-definable snap points and snap guides, as well as object axis alignment guides, all useful for the accurate positioning of objects.

Read about The PCB Editor Workspace
The PCB editor is a true, 3D design space, where the designer can easily toggle between the 2D and 3D display modes. Component models can be created in the PCB library editor from a set of simple 3D shapes, or 3D models can be imported, in a variety of formats, including STEP.

As well as importing component models, the designer can also import the product case, and 3D clearance checking can be performed (hover the cursor over the image).

For a rigid-flex design, the board can be interactively folded (as shown in the image above), ideal for performing clearance checking of the board in its installed state. The completed board can also be exported in the 3D STEP format, in its folded state if required, ready to load into your MCAD design software.

Read about The Advantage of 3D in ECAD-MCAD Integration
Design requirements are applied via design rules, using an elegant - target these objects and apply those requirements - approach. Rules are defined independently of the objects, and can be exported from one board design and imported into another. During editing or rule checking, the rules engine automatically identifies the highest priority rule that applies to each object.

A rule targets objects using a keyword-driven query language, which can range from broad identifiers, such as NetClass or All, right down to a tightly defined query that precisely targets a tricky, situation-specific, design requirement.

The PCB Rules and Violation panel simplifies the process of understanding and interpreting which objects a rule applies to, and why it is failing. Batch DRC produces a detailed report, which can be generated in a number of formats.

Read about Specifying the Design Requirements - Design Rules
Routing is no longer a simple, join the dots process. Fast device switching speeds mean that many boards have high-speed signals, requiring controlled impedance routing. The PCB editor's routing width design rule can be width-driven or it can be impedance-driven, where the routing width changes as the routing moves from one layer to another. Interactive routing is fast and efficient, with workaround, hug and push modes that let you get the job done quickly and efficiently.

There is full support for differential pair routing, as well as single-sided and differential pair length tuning.

The topological autorouter produces routes like that of a skilled board designer. Being a topological router, it is not constrained to an orthogonal grid, instead being guided by preferred direction settings and connection paths.

Read about The Routing
Generating Outputs

Ultimately, the objective of the board design process is to generate the output files needed to fabricate and assemble the board. As well as all of the popular output standards, such as Gerber X2, ODB++ and IPC-2581, you can also generate detailed drawings, and PDF files complete with bookmarks to all of the components and nets.

OutputJobs allow all outputs to be configured and generated from a single location, and the OutputJob can easily be transferred from one project to the next.

Comprehensive and configurable BOM generation capabilities are available, including ActiveBOM, which gives visibility into the project costing from the get-go.

3D PDF is also available, allowing your to share your completed PCB as a 3D PDF, complete with zoom, pan and rotation, as well as bookmarks to every component and net.

Read about More About Outputs

Where to Next?

Like all of Altium's design technologies, the PCB editor is designed to be quick to learn and easy to work in. Context sensitive right-click menus are used extensively, and context-sensitive help (F1) and in-command shortcut lists (Shift+F1) are available everywhere.

If you're new to Altium's design software, you might like to start with the concept to completion tutorial - based around a simple nine component circuit, you'll start with a blank schematic sheet and end up with the PCB, along with the files needed to fabricate the board.

Otherwise, you might like to check out the following articles:

The PCB Editor Workspace
The Board
Placing and Editing Objects
Specifying the Design Requirements - Design Rules
The Routing
The 3D Advantage
Collaborative Board Design
Finalizing the Board Design

Source URL: https://www.altium.com/documentation/display/ADES/((More+about+PCB+Design))_AD