Mixing RF, digital and analog circuits on the same PCB

A dynamic link integrates the PCB schematic and layout tool with RF design and simulation tools, resulting in a solution that overcomes the shortcomings of the classic RF design.

By Per Viklund and John Isaac

The presence of RF circuitry on printed circuit boards (PCBs) was once primarily limited to military and aerospace industry requirements. Now, the proliferation of the wireless handheld communications and remote-control devices is driving the need for mixed analog, digital and RF designs at a significantly increasing rate. Handhelds, base stations, remote controls, Bluetooth devices, computer wireless, many consumer devices, and mil/aero systems now all contain RF.

For years, RF design has been a special art, requiring specialized design and analysis tools, used by highly specialized designers. Typically, the RF portion of a PCB was designed by that specialist in a separate environment and then merged into the rest of the mixed-technology PCB. This process was highly inefficient, often required iterations to marry the mixed technologies together and resulted in multiple, unrelated databases representing the final product.

RF design paradigm has changed

In the past, design functionality was performed (and repeated) in two separate design environments through simple, non-intelligent ASCII interfaces (Figure 1). Both the PCB system design and the RF specialized design systems had their own libraries, RF design databases, and design archiving. It required that design data (schematic and layout) and libraries be managed (and synchronized) in both environments through the cumbersome ASCII interfaces.

With this old methodology, the RF designer basically was developing the RF circuitry isolated from the rest of the PCB system design. The RF portion was then translated into the PCB design using ASCII files to create schematic and physical implementation on the host PCB. If problems exist with the RF circuitry, the design must be corrected in the stand-alone RF solution and re-translated into the host PCB. The result was a total replacement versus an incremental change.

An additional problem is that the simulation in the RF solution only simulates the RF circuitry and not the surrounding PCB traces, vias, and ground planes within the RF environment. These additional shapes can have significant effects on the RF circuitry operation.

This old methodology has been used successfully for years to design mixed-technology boards, but as the RF content in products increases, the problems with having two separate design systems is starting to significantly impact designer productivity, time-to-market, and quality of the products.

With these issues in mind, Mentor Graphics has developed a dynamic link that integrates the PCB schematic and layout tool with RF design and simulation tools, resulting in a solution that overcomes the shortcomings of the classic RF design. Working with RF design experts, a set of requirements was identified and a new solution designed.

RF-aware PCB design

No integration—no matter how good—can help maintain design intent between PCB design and RF design unless there is a common understanding of the technology-specific environment between the tool sets. In other words, the typical layer-oriented structures in PCB layout has to be understood by the RF design tools and the parametric planar microwave elements used in the RF design environment must be fully understood by the PCB system.

Another key issue is that PCB systems typically regard RF shapes as short circuits and this prevents proper design rule checks (DRC) of the design. With today's complex RF system designs, functional RF aware DRC is a must to enable a correct by design methodology.

All these together contribute to the design intent. Preserved design intent is critical as this is the foundation to support multiple iterative roundtrips of design data between tools without losing information.

RF design is by nature a very iterative process. A design is tweaked or optimized in many steps. It was difficult in the past, if at all possible, to do this in the context of the real PCB design.

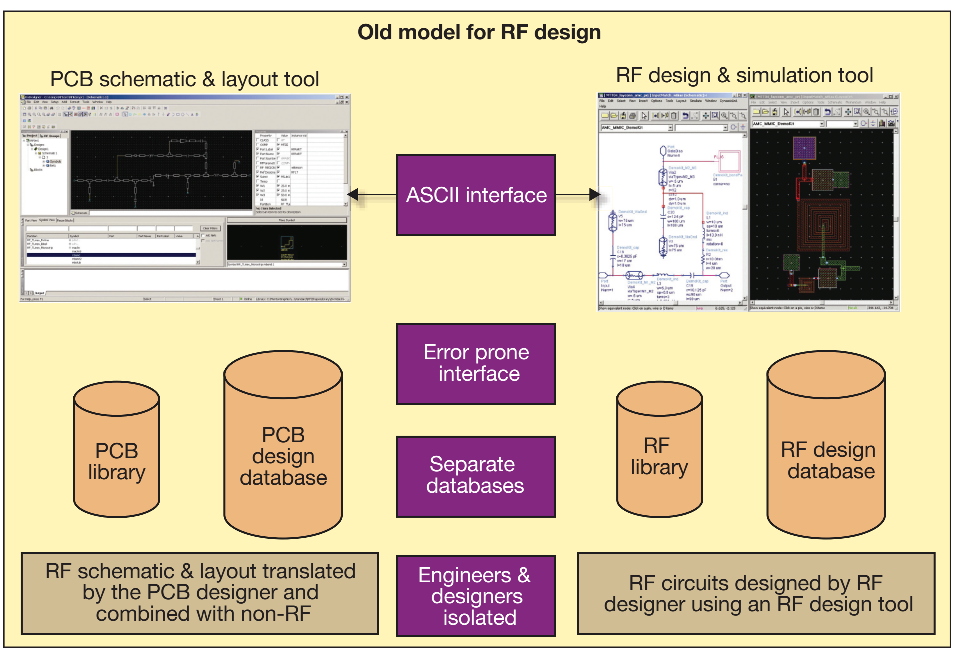

Figure 1. Separate systems interfaced with cumbersome ASCII files works but can cause loss of designer productivity and a non-competitive product.

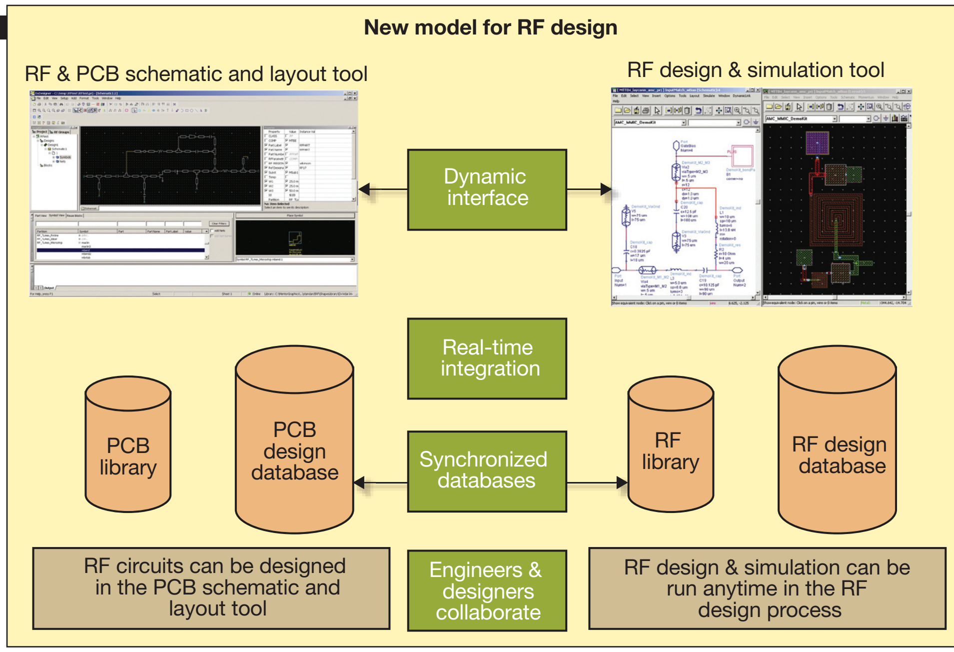

Figure 2. Tightly integrating the PCB and specialized RF systems improves designer productivity.

When the optimized RF module was implemented on the PCB, there was no guarantee that it would still work in an optimal manner. As a validation, the PCB implementation was sent to electromagnetic field analysis (EM).

There are several problems with this design flow. First, the circuit is pushed to simulation as simple metal polygons, so there is no way to modify the metal in the RF tool and send the optimized result back to PCB design and still have an intelligent RF circuit. Second, EM solutions are typically time consuming so it may be best to wait until it's really needed.

In the new flow, as the PCB and RF tools share a common understanding of the design intent, the circuit can be looped back and forth between the toolsets multiple times without loss of design intent. This means that both circuit simulation (which is very fast) and EM analysis (when needed) can be repeated and results can be compared for every change made to the circuit. Above all, this is done within the context of the real PCB with fractioned ground planes, RF shapes, traces, vias, and other components.

Libraries: Garbage in, garbage out

Libraries have always been a hurdle in RF system design. The standard components in the RF library (capacitors, resistors, transistors, etc.) frequently lack some of the parameters required for the PCB design and manufacturing processes. Likewise, the PCB design libraries usually don't contain the planar microwave elements used in the RF domain to build up RF circuitry.

In the past, a snapshot has been taken of the microwave element library, but as with any snapshot, it could be outdated in no time, forcing designers to manually ascertain that the PCB and microwave library is kept in absolute synchronization. And not just synchronized, but perfectly synchronized to ensure performance on the PCB is 100% identical to what you simulate. Obviously, as this process involved people, it failed now and then. The new integration solves this dilemma using an inter-tool dynamic link to synchronize the library.

Integrated flow

With the foundation in place, the integration between the RF design tool and PCB needed an overhaul. For more than 10 years, this integration has been based on two-way translation of ASCII IFF-format files. Although, capable of holding a portion of the design data, this format is far from adequate to support seamless round-trip integration. Lack of library synchronization is one of the more critical issues. RF and board designers have struggled with this model for along time and despite several attempts to improve the interfaces only marginal results were seen.

Something different had to be developed and this led to a network-based inter-tool communication providing a dynamic two-way link between RF design and system-level PCB design (Figure 2). To fully support today's concurrent engineering processes, multiple board designers can operate simultaneously on the same design database and each link to one or multiple simulation sessions. Now an RF module can be designed in the RF design tool and, when appropriate, be pushed over and become an intelligently integrated part of the system-level schematic and PC board rather than the dumb black box circuit of the past. At this stage, updates to the circuits can be made in either environment and the impact be simulated.

Each RF circuit is contained as a grouped object to help maintain traceability, version management and design reuse. As design intent is preserved, any number of iterations can be processed without the usual cost in cycle time. In addition, as the RF module can be simulated within the context of the actual system-level PCB, its function can be validated at a more detailed level to help cut design cycles.

RF PCB bottlenecks

There are several well-known RF PCB design bottlenecks. First, as each RF module on a board may have been designed by a separate RF design team and the module may

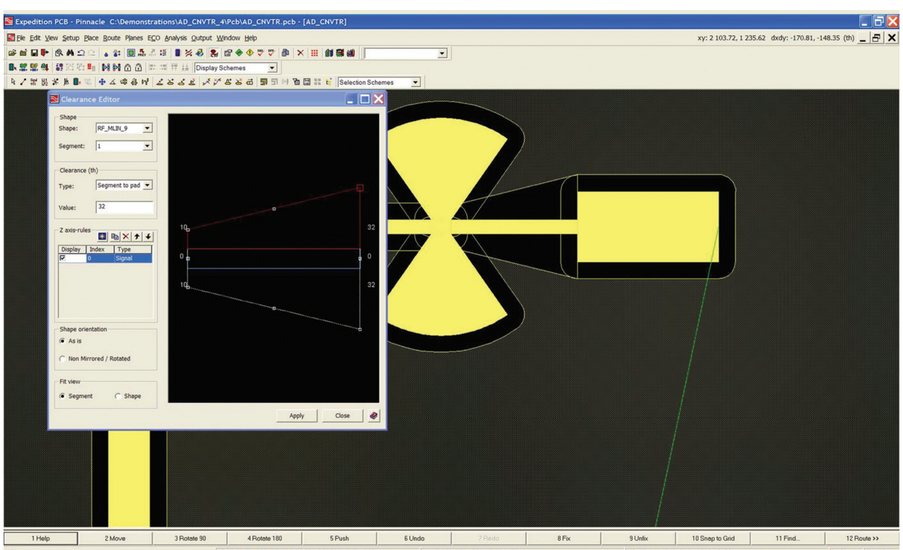

Figure 3. Because of the highly integrated interface, parametric RF shape clearance on the PCB design is precisely as the RF engineer defined it.

live its own life in terms of versioning, variants and reuse, it becomes vital to be able to manage the circuit as a group that can be managed as one entity and its origin and version be traced—but still be accessed as individual circuit elements at any time. To resolve this issue, the schematic and layout tools were expanded to support hierarchical circuit grouping. This way, even though an RF circuit is laid out on a PCB, it is still kept together as an RF circuit and can be linked to the proper RF team for analysis.

The next hurdle is ground plane clearance. In the classic design process, the RF metal was imported as a black box piece of metal and ground clearance was handcrafted as plane voids on every layer needed. When the RF circuit was updated—which was a frequent operation—the cutouts had to be manually edited to reflect the new circuit. This edit process alone can take weeks for some designs.

With a new design flow that truly promotes iterative updates between RF design and PCB design; manual updating is way too slow. Instead, an intelligent parametric RF shape clearance is introduced to let the RF circuit clear ground exactly the way the RF engineer defines it, and to have it parametrically updated as the RF circuit evolves during design, as shown in Figure 3. This parametric plane clearance cannot only be defined for the same layer on which the RF shape is placed, but also for layers above and below the shape, including the solder mask. If the RF circuit is updated with changed dimensions or if it's being moved to a new layer, these cutouts automatically update, saving a tremendous amount of cycle time.

Interconnection between RF elements on the PCB typically uses meander lines instead of normal PCB traces to connect RF circuits. These meander lines can have tapered width changes, optimal impedance miter, or curved bends.

In the past these were made as metal plane shapes and were difficult to edit. Furthermore, as they were just metal polygons, the only way to simulate was to use a time-consuming EM solution. Mentor has solved this dilemma by designing a new meander line design object for its PCB tools. This way, the PCB designer can connect RF signals effectively and when simulation is needed, the meander lines can either be sent to EM analysis—as in the past—or automatically be decomposed into fast circuit models.

A striking feature of most RF system designs is the very large number of via holes stitched along RF shapes, around plane contours or peppered over plane surfaces.

This so-called via stitching is used to reduce radiation losses when stitched along RF shapes or when peppered across planes, to prevent parallel plane excitation.

Adding these vias manually costs countless hours—or days—and need manual rework each time the circuit is updated in design iteration. Many board designers developed smart scripts and programs to add the vias but the issue with rework is still unsolved.

Now, designers can automatically generate via patterns and contour stitching parametrically in elaborate patterns, multi-row stitching along shapes, and even include them in the EM simulation, as shown in Figure 3.

Conclusion

The new RF design paradigm has slowly put RF design companies in a tricky situation with unacceptable cycle time and excessive design cycles. We are now working together with RF tool vendors at a completely different level than what has been the norm for the past years in order to provide a design flow that is tailored to meet the challenges seen in the industry today and in the future.

The prime goal—to cut design cycles—is reached by ensuring a synchronized library and by facilitating a fast and easy integrated simulation flow. As designers can simulate frequently as the design evolves, the system can be validated up front. RF-aware DRC promoting correct by design also contributes.

Cycle time was traditionally wasted in cumbersome file translation between tools and in the fact that the PCB tools did not understand RF or even support some of the primary RF design requirements.

It's safe to say that ASCII file transfer is a relic of the past. The demand for integrated design teams across technology and global boundaries dictates direct tool integration where the tool sets share a common understanding of RF. RFD

ABOUT THE AUTHOR

Per Viklund is director of RF and IC Packaging at Mentor Graphics' Systems Design Division.

John Isaac is marketing development manager for Mentor Graphics' Systems Design Division.