Beyond Design: Top Gear – PADS Professional Road Test

★★★★½ Rating: 4.5

by Barry Olney | In-Circuit Design Pty Ltd | Australia

We hear all the hype about new EDA tools but how do they actually perform on your design? This month Barry Olney road tests Mentor Graphics' new PADS Professional and puts it through a rigorous performance evaluation – let's see how the Xpedition technology actually performs integrated into the PADS tools.

Opening the hood, we see an impressive line-up of features including Signal and Power Integrity, Thermal Analysis and DRC support for traces violating split planes, reference plane changes and shielding. All the essentials for today's complex high-speed designs! Plus, I am looking forward to trying the dynamic plane generation feature – regenerating copper pours is always a pain in any software. And, of course, PADS Professional includes all the standard features one would expect in a high-end tool.

Based on Xpedition technology, PADS Professional is a major improvement from the previous PADS suite of tools. I was first impressed by this technology way back in 1994 when I attended VeriBest PCB training and the sales kick-off in Boulder, Colorado. During the sessions, a few of the Intergraph Electronics sales guys were taken out the back, into R & D, and where shown the latest routing technology – eyes lit up with dollar signs as the VeriBest (now Xpedition) router was put through its paces.

Meanwhile back in Australia, where I was responsible for Intergraph Electronics sales and support, customers were also suitably impressed. My first sale was six seats of VeriBest PCB with 20 seats of Design Capture to Fujitsu Australia, who previously used Cadence. Both Cadence and Mentor presented their flagship products Allegro and Board Station but the VeriBest router was so impressive that the competition did not rate mentioning. Ron Oates, CAD Manager, Fujitsu Australia [at the time] stated in a press release: "VeriBest is light years ahead of the competitors". And, it still is arguably the best routing technology available today. Mentor went on to acquire VeriBest Inc, in 1999, as the lack of routing technology formed a fairly large hole in their PCB offerings. Needless to say, Mentor's stock rose 9% after the acquisition was announced.

I won't bore you with a full list of functionality or standard EDA tool features but rather I will take you through, in detail, what I see as the outstanding features of PADS professional.

PADS Professional utilizes xDX Designer as the front-end design entry tool. This schematic capture package was originally called ViewDraw from ViewLogic and became the unified front-end tool for all Mentor PCB products some years ago following an acquisition. Originally developed for creating Hardware Description Language (HDL) function blocks for digital and mixed signal systems, such as FPGAs and ASICs, it has a multitude of interfaces and is adaptable to many environments. In the PADS environment, it interfaces to the PCB (of course) but also allows FPGA I/O Optimization, the integration of library tools, DxDatabook and downstream digital and analog (EZWave) simulation tools.

But, the ability to launch HyperLynx LineSim at schematic level is its best attribute as far as I am concerned. After selecting a net, the LineSim link loads the data from xDX Designer and exports it to HyperLynx to create a pre-layout free-from schematic view of the nets topology as in Figure 2. You can then simulate a sample of nets for say data, address, clocks and strobes to define the layout design rules. These rules are embedded, in the schematic, via the Constraints Manager and will then flow through to the layout database with forward annotation. Constraints are maintained through a

common database that is consistent and in an easy to use spreadsheet interface. There is no need to learn an obscure program language to create complex constraints as in other tools.

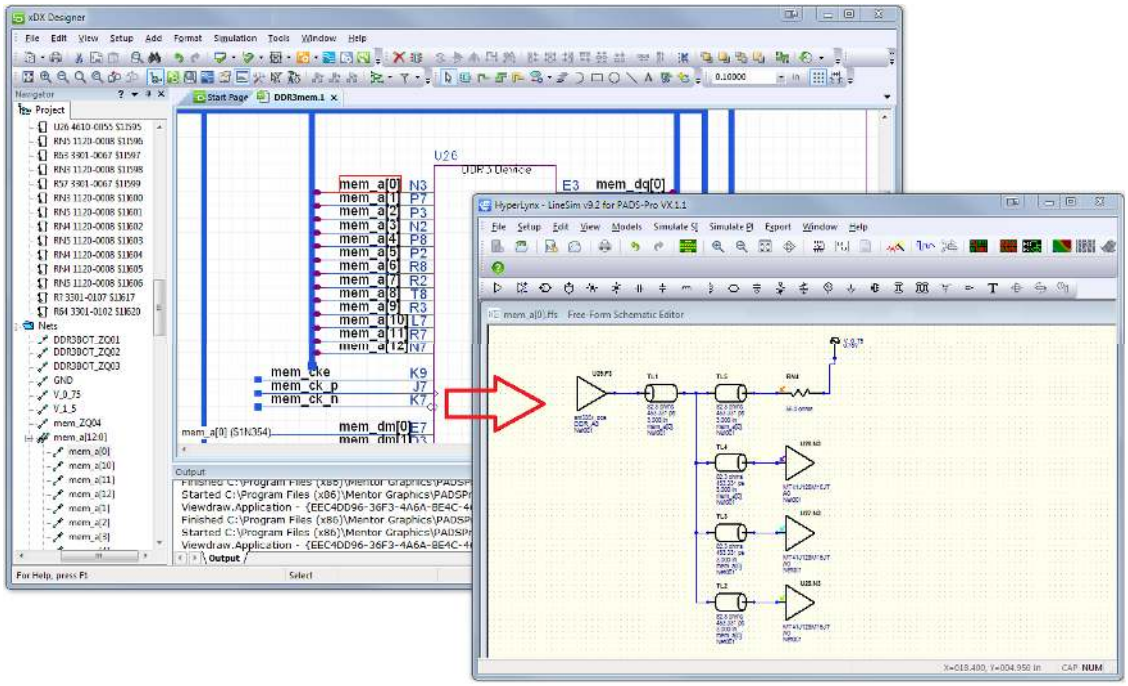

Figure 2 – DDR3 memory address net topology exported to LineSim.

Consequently, any unsightly waveforms (under/overshoots) can be dealt with by the Termination Wizard, which automatically selects the correct type and value of termination based on the IBIS model characteristics and that of the transmission line. It really makes signal integrity too-easy. And, providing engineering intent, to the layout designer, at the schematic level imparts an error free process.

Integration of the schematic and PCB is through a common database so there are no netlist errors to contend with. Simply package and begin layout. The first thing I noticed, when I entered the PCB environment is the smooth panning of the graphics. Once I got used to the correct mouse buttons to use, panning and zooming were a breeze – amazingly fast and smooth. Another highlight is that if you happen to move a previously routed component – it just re-routes the connections for you. Obviously this has limitations, such as large BGAs, but most ICs can easily be moved to rearrange the placement and routing. Moving vias and routed traces is also a breeze. Simply click and drag, with seemingly limitless shove capabilities, and even differential pairs are pushed and shoved whilst maintaining their design constraints.

Additionally, the use of mouse strokes, for quick execution of commands, rather than hotkeys is a pleasant flash-back from my UNIX past. Basically, you can define stroke patterns to represent any command so that you just keep moving the mouse rather than having to refocus on the keyboard. This makes executing commands very fast.

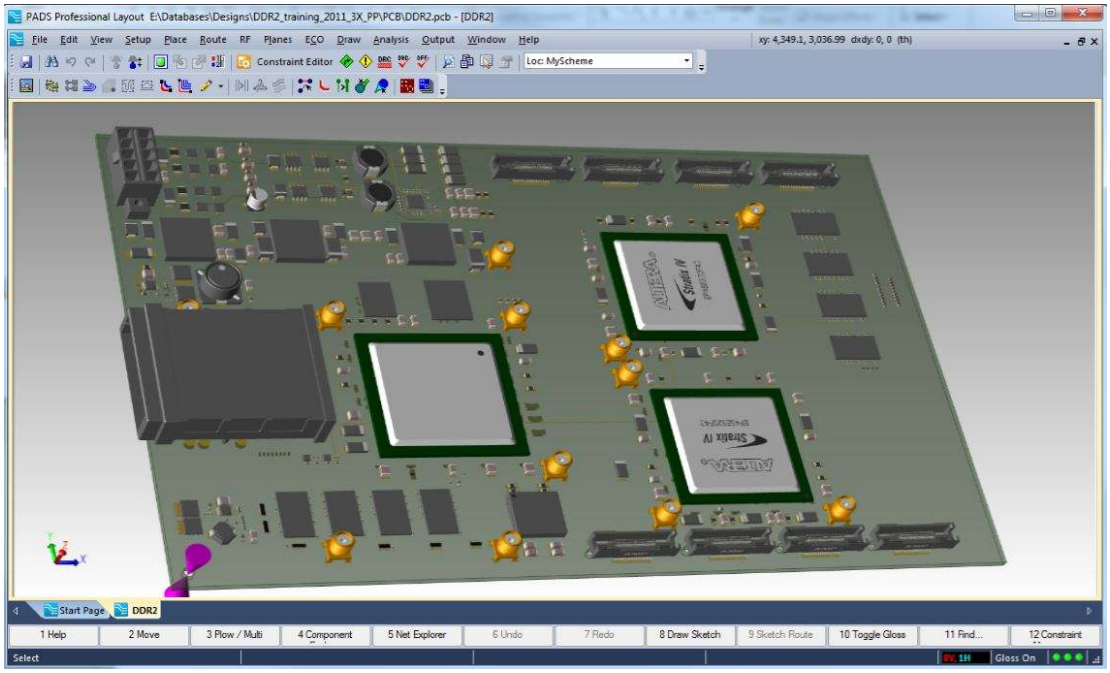

Figure 3 – 3D view of the PCB

As with all Mentor PCB design tools, the ability to place and route directly from the schematic is a major time-saving trait. The Component Explorer is used to setup placement. You can place parts directly from the select list or by cross-probing with the schematic. The latter is the most effective way, as the components are then placed by functionality. Placement groups of components can also be defined. Components can also be placed in the 3D view, which gives you added confidence in the avoidance of height restrictions and interference with other components. A cut in 3D can be placed to view the cross-section. Additionally, a 3D mechanical assembly (such as an enclosure) can be imported to enable the visualization of the overall product fit. Exchange of data with MCAD tools is achieved via the collaboration tool. The obvious downside here is that this is not 'live'. A COM Server dynamically linking ECAD/MCAD applications would be a more elegant solution for real time collaboration. But, the collaboration tool would work well in a large organization where an MCAD department has to sign-off on the mechanical aspects.

Are you tired of shelving and repouring plane areas? – I certainly am! But what if you could simply plow straight through a copper pour or plane and gracefully push the copper out of the way as you go? The dynamic plowing is as smooth and fast as routing a normal trace. As the trace plows through the copper, clearances are effortlessly generated automatically.

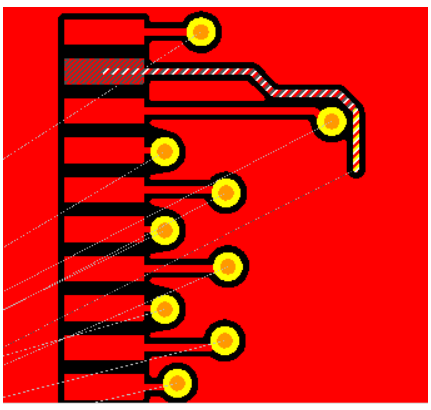

Figure 4 - Plowing dynamically straight through a copper pour

PADS Professional provides a selection of routing tools with each optimized to perform a particular function. Typically, using one routing tool will not satisfy the requirements of an entire design. Obviously, you will not achieve acceptable results if the entire design is completely autorouted. The typical flow would be to first setup the design constraints, fanout, interactively route and tune critical nets and then use the automated tools to perform the more mundane tasks including clean-up after fixing critical signals.

But once you get your hands on the router – WOW – what can I say! You certainly won't want to digress back to that inept PCB tool you previously used with pride. This routing technology is definitely something you have to see to believe. For years, it has left rivaling tool salesmen scratching their heads with disbelief and it just keeps getting better.

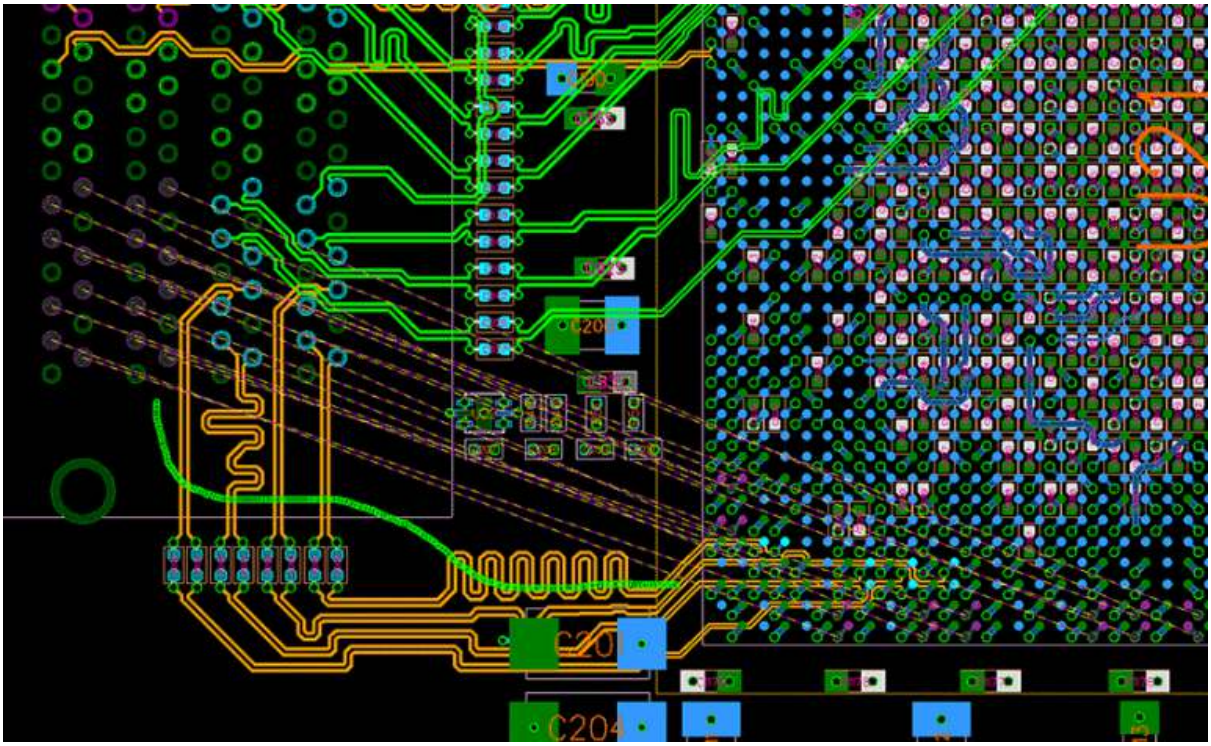

Figure 5 – Interactive Sketch Routing technology

The latest evolution of the Mentor's routing technology is the 'Sketch Router'. There are two factors that enable high performance – escape optimization and multiple net routing. When considering the set of netlines to route, the sketch router will simultaneously escape the two ends, of the netlines out of the BGAs, and order them such that the routing can be completed without additional vias. It is this approach that makes an incredible difference in performance. There are two modes to sketch routing – packed or unpacked. The packed style groups the traces together and is very useful for dense, synchronous buses. The unpacked style uses a more direct and efficient approach that will naturally spread the traces apart reducing crosstalk between adjacent trace segments. In both cases, the tuning algorithms are post-processed with the existing traces pushed and shoved to allow space for the serpentines. Screenshots of the router really don't do it justice. I suggest you view some of the PADS demo videos to get a feel for how it works dynamically.

Once the board is completely routed, and you are satisfied with the tuning, HyperLynx BoardSim can be launched to verify the signal timing and analyze any crosstalk that may be produced from adjacent signal segments. A preliminary generic batch simulation scans nets on an entire PCB, flagging signal integrity, crosstalk and EMC hot spots. Then interactive simulation takes the analysis to the next level – simulating trouble spots identified by the batch analysis in order to further resolve the issues with

greater accuracy. Field solver views clearly show the adjacent or broadside coupled crosstalk regions. A thermal simulation can also be run to check for hot spots. A DDR batch mode wizard, HyperLynx DRC, for SI rule checking, and DC drop analysis are also available as options. This is an impressive line-up of post-layout simulation tools.

In conclusion, PADS Professional certainly lives up to all the hype. The latest routing technology is fast, smooth to drive and hugs the corners well. With all the horse-power you need for the most demanding design, PADS Professional will definitely get you across the finish line ahead of the field. I will give it a 4.5 star rating simply because nothing is perfect – although it doesn't get any better than this.

References

Mentor Graphics PADS Professional documentation
http://www.eetimes.com/document.asp?doc_id=1140801
For information on the PADS Professional, please go to www.pads.com/professional

Bio:

Barry Olney is Managing Director of In-Circuit Design Pty Ltd (ICD), Australia. The company developed the ICD Stackup Planner and ICD PDN Planner software, is a PCB Design Service Bureau and specializes in board level simulation.