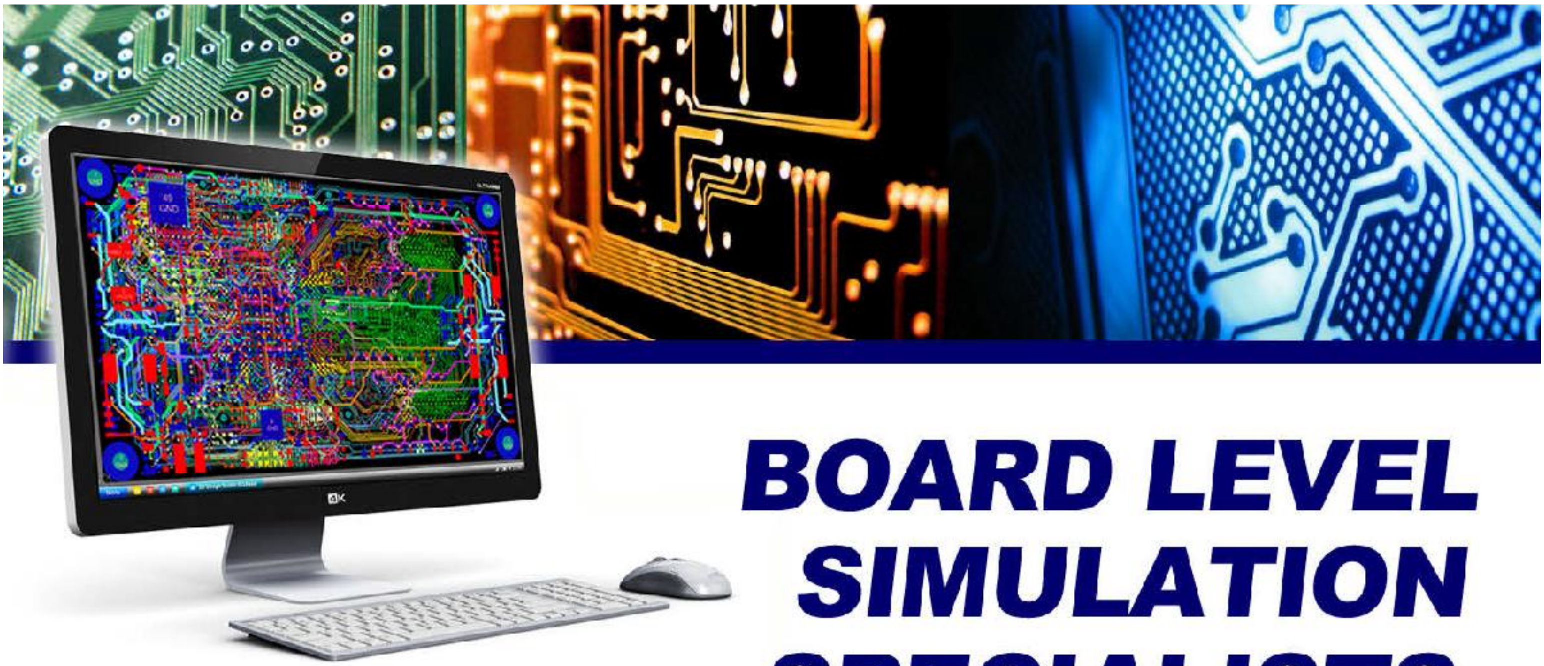

BOARD LEVEL SIMULATION SPECIALISTS

ICD Stackup Planner - offers engineers/PCB designers unprecedented simulation speed, ease of use and accuracy at an affordable price

- 2D (BEM) field solver precision
- Characteristic impedance, edge-coupled & broadside-coupled differential impedance
- Unique field solver computation of multiple differential technologies per stackup
- Heads-up impedance plots of signal and dielectric layers
- User defined dielectric materials library - over 28,000 materials up to 100GHz

ICD PDN Planner - analyze multiple power supplies to maintain low impedance over entire frequency range dramatically improving product performance

- Fast AC impedance analysis with plane resonance
- Definition of plane size/shape, dielectric constant & plane separation for each on-board power supply
- Extraction of plane data from the integrated Stackup Planner
- Definition of voltage regulator, bypass/decoupling capacitors, mounting loop inductance
- PDN EMI Plot with EMC Limits. Frequency range up to 100GHz
- Extensive Capacitor Library – over 5,650 capacitors derived from SPICE models

iCD

In-Circuit Design Pty Ltd

LEARN MORE

www.icd.com.au

The Case for Artificial Intelligence in EDA Tools

by Barry Olney
IN-CIRCUIT DESIGN PTY LTD / AUSTRALIA

I-Connect007 Editor Andy Shaughnessy reported that the keynote speaker at the IPC APEX EXPO Design Forum was Dale Parker, a former PCB designer at Shure who is now a CAD manager at Google X. Parker is involved in the development of autonomous vehicles and all sorts of other great ideas at Google X. According to Andy, Parker told the crowd, among other things, that EDA tool vendors need to trash all their old 1990s code and start over, this time with artificial intelligence.

There has been a lot of activity in the field of AI recently, with such developments as voice recognition, unmanned autonomous vehicles and data mining to list a few. But how could AI possibly influence the PCB design process? This month, I will take a look at the endless possibilities.

So much time is wasted on reproducing the same thing over and over again on each layout. Current EDA tools, with all their bells and whistles, are still very limited in automation processes and mostly rely on the skills and foresight of the engineer and PCB designer to drive the software through all the hoops. Instead, EDA tools need to predict what the designer is trying to do, then look at previous designs to suggest alternatives and auto-complete the design where possible. AI is a system that perceives its environment and takes actions to maximize its chances of success.

Automating many of the tedious steps in setting up the initial database would be a good start. A standard form factor could be used to establish the initial layout environment ensuring that designs are compatible across multiple generations of technology. Although some PCB layout tools allow the designer to load a standard set of predefined startup configuration files, there is still too much manual intervention required. The PCB database could predict the fundamental design rules and via stack requirements sourced from previous experience.

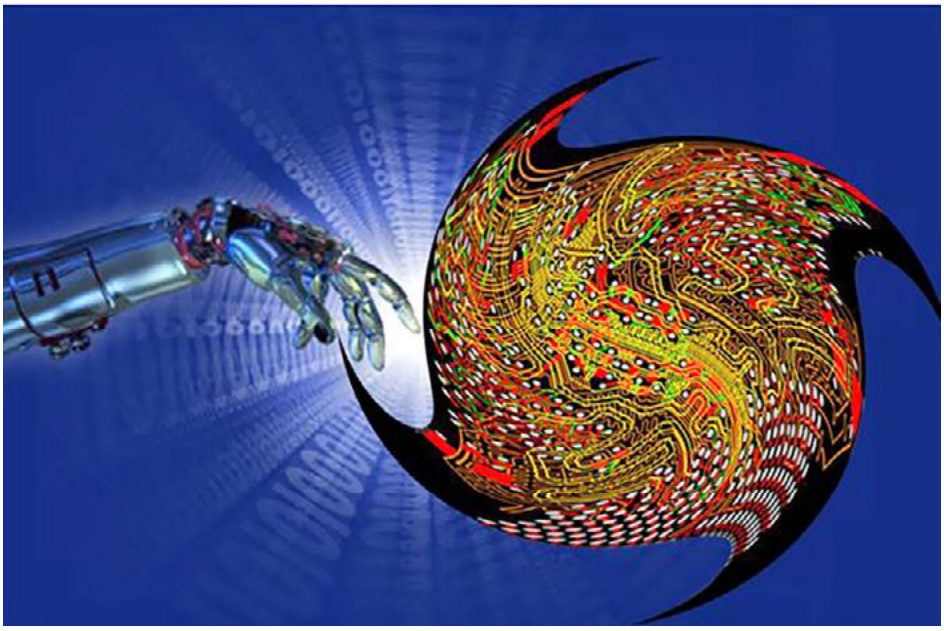

Figure 1: Artificial intelligence is part of the future of EDA tools.

Predictive text, which we all use every day on our cell phones, could provide self-evident naming conventions for supplier part numbers and database fields, greatly speeding up the design definition. Busses and interfaces could be analyzed and categorized with naming conventions interpreted from the chip pin name assignments, eliminating much of the monotonous schematic capture process. IC power pins could have powers supplies assigned based on datasheet requirements. And a starter set of decoupling capacitors, added to each power pin, could kick off the PDN analysis based on previous capacitor availability and parameter selections.

A selection of predefined library components could be offered, based on an initial bill of materials, and pre-placed on the schematic predicting the designer's requirements. IBIS models could be automatically assigned to each chip, based on the part number and all the interconnecting transmission lines identified. The IBIS model's source and load impedances could be extracted to assign the required impedance and terminations to each individual transmission line.

Also from this, the board stackup could be created based on previous designs, with similar technology, selecting dielectric materials, from a well maintained library, sourced from the preferred fabricator availability, dielectric loss and bandwidth requirements. Data and address busses together with clock/strobe different pairs, defined at the schematic entry level, could be assigned to certain layers in order to minimize crosstalk, electromagnetic emissions and return path loops. Power plane shapes could be automatically defined based on component placement and on the pins that need to be connected, allowing for DC drop and maximum current supply.

Memory blocks—whether they be synchronous, asynchronous, source synchronous, clock-forwarding or embedded clock—could be recognized and standard design rules deployed. Busses and interfaces could be also analyzed and recognized. The entire design rule set could be built from a combination of these requirements and those learnt from previous similar designs.

Functional blocks that the designer is working on, could be scrutinized in order to anticipate which blocks might be useful in future designs and these could then be made available to other designers on the corporate intranet. A database of reusable placement and routing blocks could be made available so that the intelligent database can readily identify a suitable block to drop into the design–the selected block could then be automatically adjusted to the specific needs of each instantiation.

Intelligent forward and back annotation would be a definite godsend. Traditionally, the schematic capture and PCB layout software were developed as separate applications coupled by the annotation process. Why can't we have a common database for both schematic and PCB totally eliminating the constant need to update in one direction or the other? When an ECO is implemented on the PCB, the schematic should instantly know and understand the changes and vice-versa. Over the years, forward and back annotation has been one of the most frustrating and time-consuming issues.

The intelligent database could suggest placement of critical components based on the established design rules, matched delays of busses and define routing strategies based on the technology used. But, will autonomous routing ease PCB gridlock? Placement changes could be

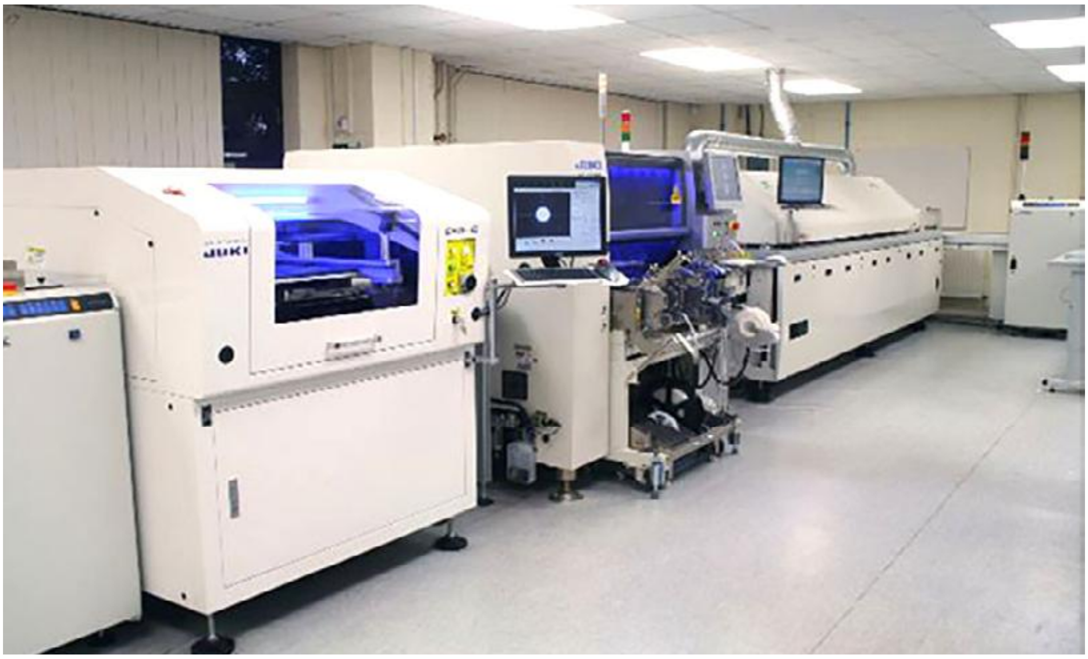

Figure 2: SMT assembly production line (Courtesy Juki).

made to open routing channels and land sizes adjusted automatically to ease routing based on design rules. The AI router could decide which layer and direction to route a bus in order to alleviate bottlenecks that generally occur in the center of the board.

The PCB project deliverables such as Gerber, IPC-2581B or ODB++ and pick-and-place files could be automatically generated based on the established design constraints. Fabrication documentation could also be auto-completed based on established standards.

On the assembly side, Internet of Things (IoT) manufacturing is currently being deployed. This system supports live bidirectional data flow between all electronics manufacturing shop floor machines and processes creating a "smart" factory with "plug and play" deployment. This allows an organization to overcome the bottleneck of establishing efficient machine-to-machine and machine-to-human communications.

I'm sure both you and I could come up with many more ideas to intelligently automate the PCB design process if time permitted. And although such a tool is many years off, the concept behind the EDA AI tool looks promising. PCB designers can certainly use all the help they can get. And, having the design tool intelligently automate their work flow could substantially speed time-to-market.

However, there is already considerable resistance to AI with some suggesting it many create too many erroneous results to be useful, and cannot be trusted. There is also the potential downside that AI might work too well and reduce the skill level needed for the role of the designer to simply indicating the goal and examining the results—but that is a long way off!

Points to Remember:

• Dale Parker, CAD manager with Google X, said in his Design Forum keynote that EDA tool vendors need to trash all their old 1990s code and start over, this time with artificial intelligence.

• Current EDA tools, with all their bells and whistles, are still very limited in automation processes and mostly rely on the skills and foresight of the engineer and PCB designer.

• Automating many of the tedious steps in setting up the initial database would be a good start.

• Predictive text could provide self-evident naming conventions for supplier part numbers and database fields greatly speeding-up the design definition.

• A selection of predefined library components could be offered, based on an initial bill of materials, and pre-placed on the schematic predicting the designer's requirements.

• The IBIS model's source and load impedances could be extracted to assign the required impedance and terminations to each individual transmission line.

• The board stackup could be created based on previous designs with similar technology.

• Functional blocks that the designer is working on, could be scrutinized in order to anticipate which blocks might be useful in future designs.

• Intelligent forward and back annotation would be a definite godsend.

• Placement changes could be made to open routing channels and land sizes adjusted to ease routing based on design rules.

• The project deliverables such as Gerber, IPC-2581B or ODB++ and pick and place files could be automatically generated based on the established design constraints.

• IoT manufacturing is currently being deployed creating a "smart" factory with plug and play. **PCBDESIGN**

References

1. Richard Quinnell, AI for software development, www.edn.com.

2. Mentor, Valor software documentation, www.mentor.com.

3. The ICD Stackup and PDN Planner software can be downloaded from www.icd.com.au.

Barry Olney is managing director of In-Circuit Design Pty Ltd (ICD), Australia. This PCB design service bureau specializes in board-level simulation, and has developed the ICD Stackup Planner and ICD PDN Planner software. To read past columns, or to contact Olney, click here.