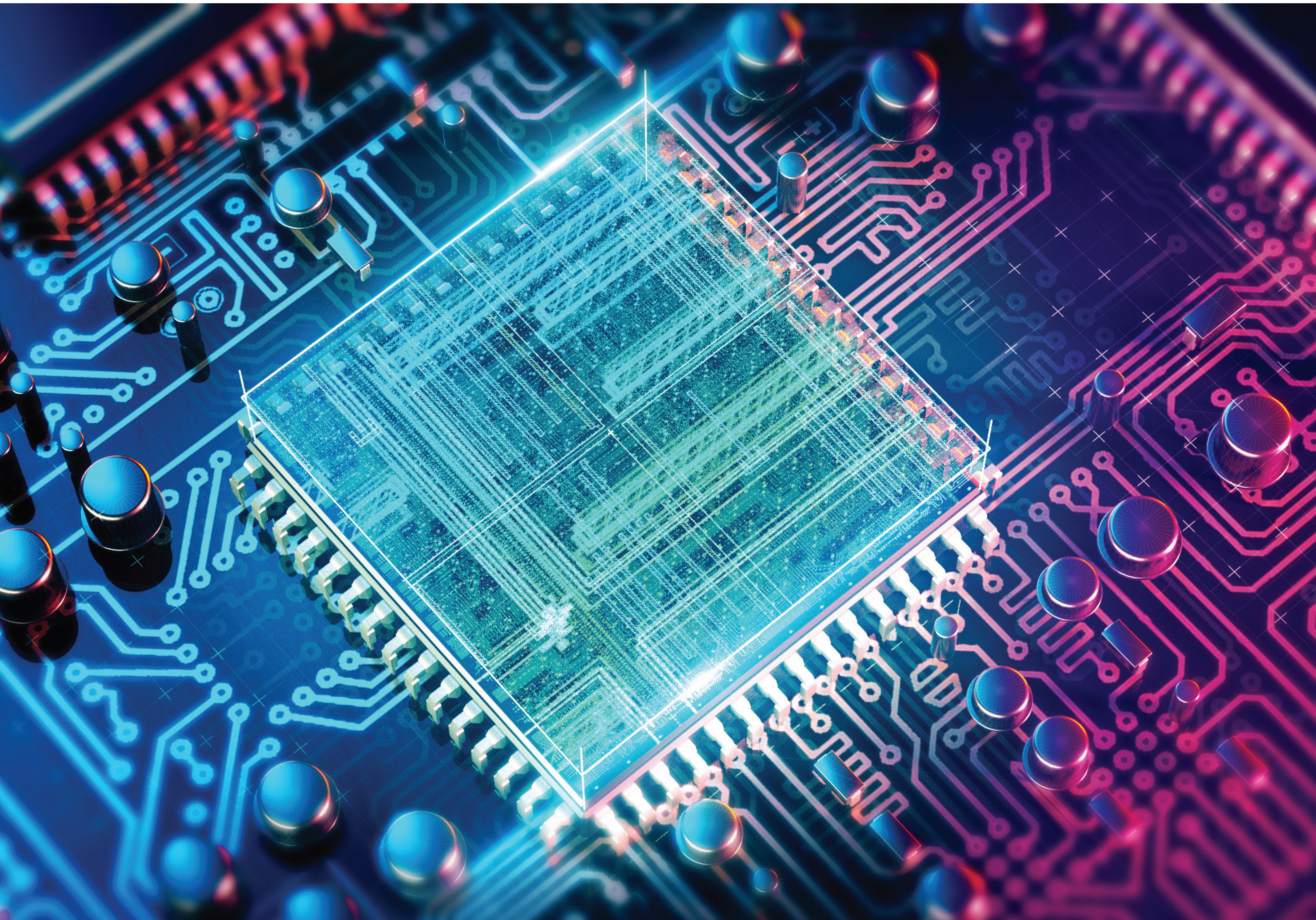
SIEMENS

DIGITAL INDUSTRIES SOFTWARE

Best practices in PCB design

The 5 pillars of digital transformation for electronics systems design

siemens.com/eda



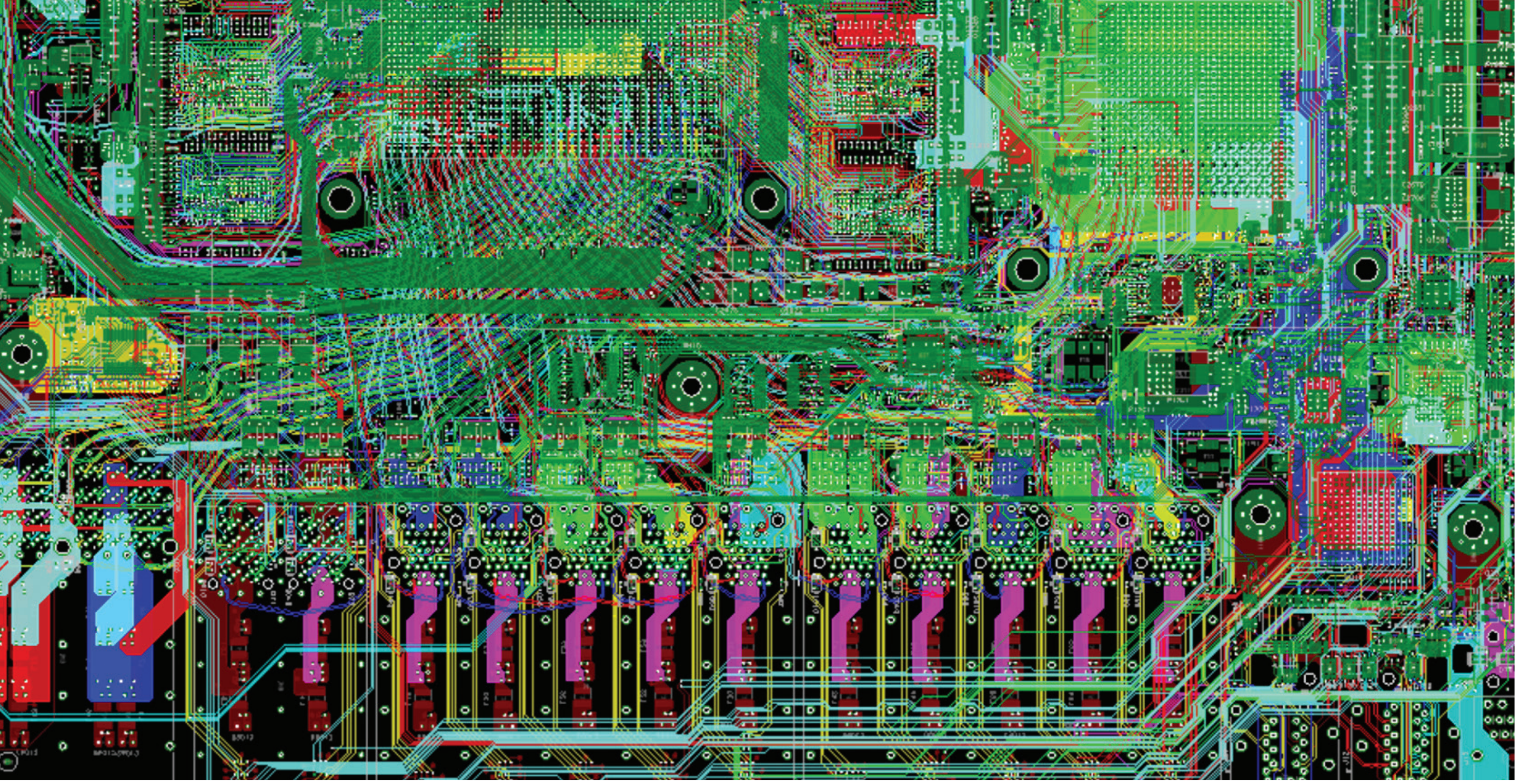
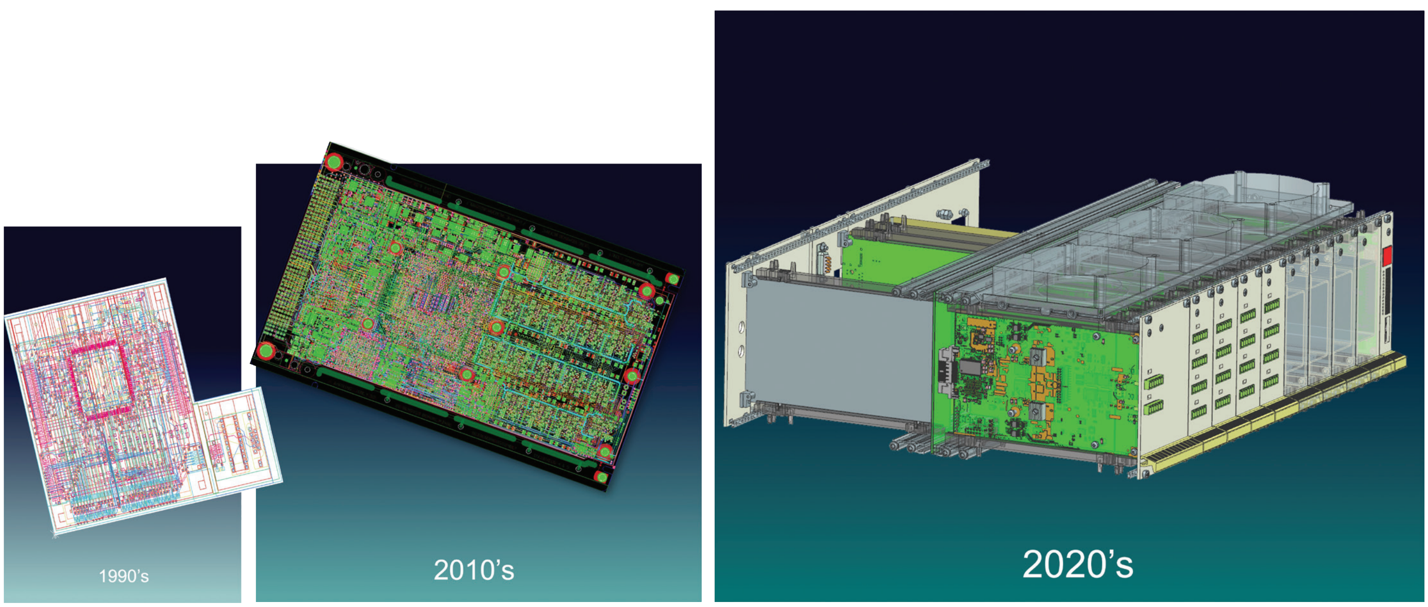


Table of contents

The growing complexity of electronics design: from single PCB to systems design

PCB design encompasses the complete journey of developing a Printed Circuit Board (PCB). It involves seamless cooperation and synergy across various domains, including electrical, mechanical, software, system, testing, and manufacturing.

In today's electronics landscape, design complexity has reached unprecedented levels. Design teams face the challenge of delivering intricate products within tight timelines. However, they often encounter inefficiencies that consume precious time. To address this, fostering collaboration across all engineering domains and providing top-tier solutions is essential for successful electronic systems design.

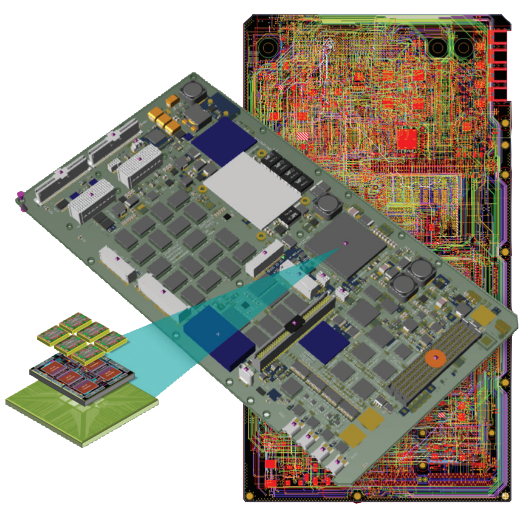


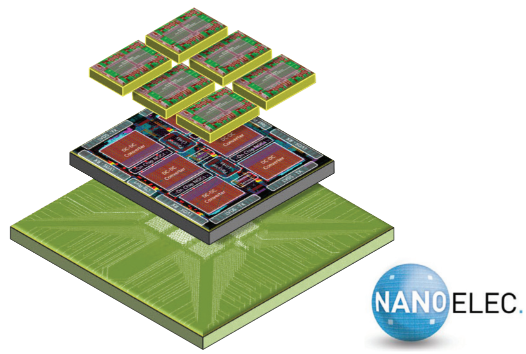
Is your company equipped to handle complexities in electronics systems design?

In the ever-evolving landscape of electronics systems design, companies face a myriad of challenges. From intricate product specifications to organizational dynamics, the complexities are manifold. Let's delve into these multifaceted dimensions:

Product complexity
- Advanced packaging technologies
- Transmission speeds
- Multi-discipline co-design
- Component count & density
- Form-factor
- Material technologies

Organizational complexity
- Collaboration across engineering disciplines
- Collaboration across geographies
- Knowledge management





Source: European CATRENE project

- Workforce productivity & efficiency
- Collaboration across different domains
- Collaboration with external stakeholder (Fabrication and Assembly)

Process complexity
- User-experience
- Integrated verification
- Heterogenous tool chain
- System engineering
- System decomposition & verification
- Streamline design to manufacturing

Supply chain complexity
- Global supply shortages
- Contractor management & assessment
- Alternate supply
- Pricing volatility
- Risk assessment
- Sourcing visibility & knowledge
- Supply chain intelligence at the point of design

Digital transformation combines all aspects of product design, enabling companies to effectively address increasing complexity in the electronics systems design ecosystem.

How are these complexities affecting teams?

Internal workflow

- Insufficient requirements lead to added, unbudgeted design hours
- Lack of resources forces double duty and increases the workload
- Dated communication and process methodologies cause delays in delivery and increase risk

Customer relationships

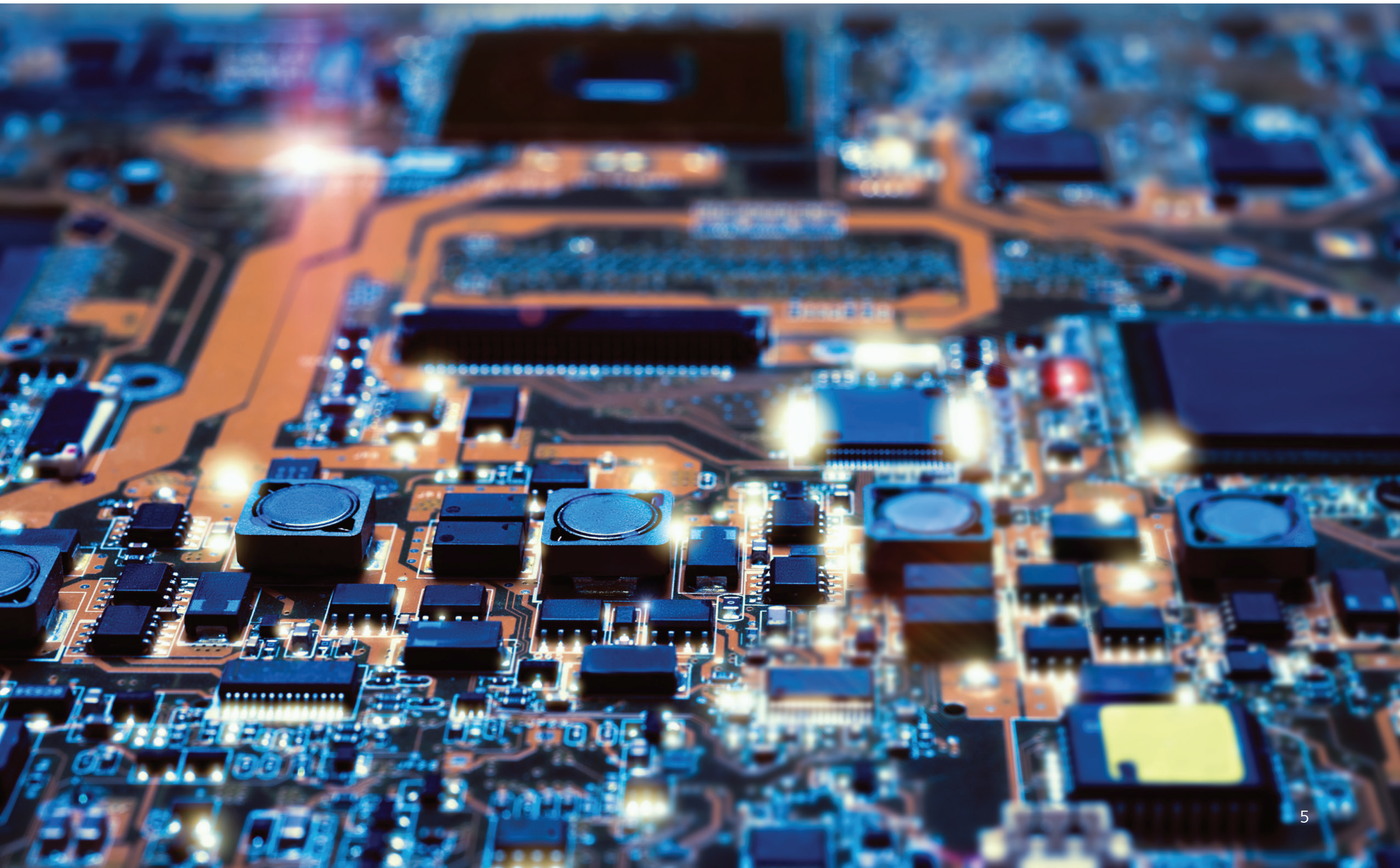
- Customers' unrealistic expectations – in terms of cost, time and quality – creates severe stress
- Out of scope changes have ripple effects for designers, such as lowered morale and burnout
- Unsatisfied customers can tarnish the company's reputation, and lead to loss of future PO's.

Supplier relationships

- DFM issues occur when suppliers are not engaged throughout the process
- Without the suppliers' expert input, the end product's quality may be jeopardized
- Suppliers can be inflexible and unwilling to work to meet the original project timeline

Best practices are standards or guidelines that are known to produce good outcomes if followed.

By implementing the 5 pillars of best practices in PCB design, companies can optimize processes, improve team collaboration, reduce design time, engineering costs and respins, and increase reliability and quality of designs.



Digitally integrated and optimized

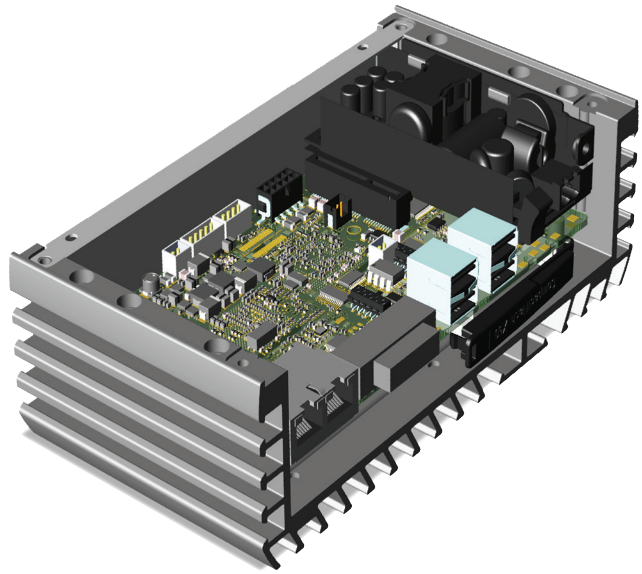
A digitally integrated and optimized multi-domain environment can enable efficient, secure, concurrent design across all engineering teams.

ECAD/MCAD co-design

Cross-domain collaboration requires ongoing, effective communication. Electrical & mechanical design teams need to know what changed, when and why – in real time.

Collaboration with MCAD systems
Communicate design data to mechanical CAD systems using a ProSTEP-approved data exchange standard. Designers can communicate between disciplines at any time or frequency, keeping participants in their respective system's comfort zone – no need to learn new tools.

3D visualization
Visualization of the PCB and its enclosure within the layout environment allows designers to optimize the layout based on mechanical and manufacturability constraints.



Preview proposed changes
To clearly identify a proposed design change, designers highlight the proposed ECAD/MCAD collaboration object within the 3D viewer before applying the modification in their design. Consistent, iterative communication ensures multi-discipline and multi-domain team synchronization.

Cloud-enabled remote collaboration
Share lightweight representations of design data and other files with colleagues and remote stakeholders in a secure cloud workspace. Powerful viewers make it easy to visualize and markup files and engineering models to help keep extended teams aligned.

Design and manufacturing collaboration

Technical Queries (TQs) have ripple effects in terms of time and cost. Most of them can be avoided without leaving the design tool or halting the design process.

Concurrent DFM analysis saves time and cost

The further into the design process a problem is discovered, the more it costs to rectify. Running concurrent DFM analysis within the design tool to locate problems allows for instant solutions during the design process.

Automated DFM analysis

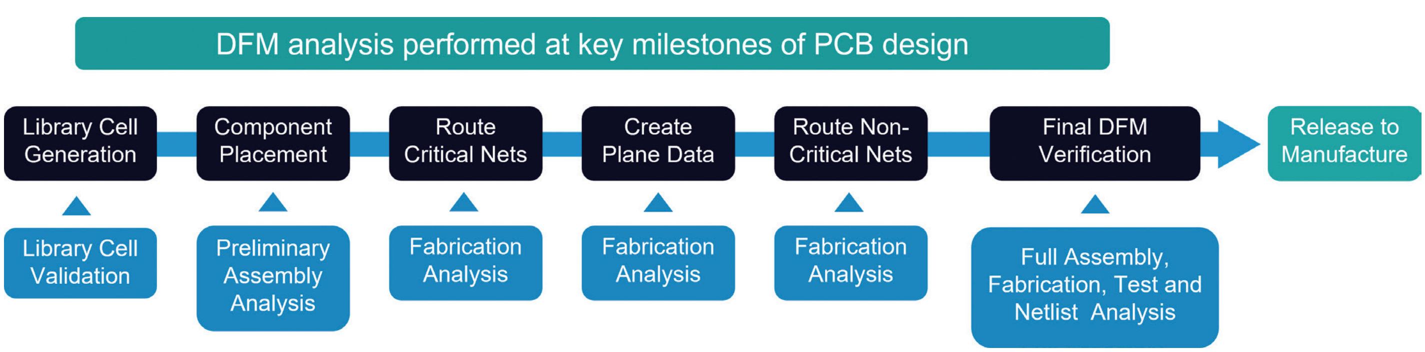
High-quality automated DFM analysis is based on a combination of PCB technologies and manufacturing constraints. Letting DFM software act on each of these constraints automatically saves engineering effort and delivers the desired results consistently for every user, regardless of their manufacturing expertise.

DFM analysis for flexible and rigid-flex circuits

Nearly 30 percent of all electronics companies have flex and/or rigid-flex circuits within their product portfolios, and they require different materials and manufacturing processes. DFM analysis technology supports these different needs.

Accurate and validated data for PCB fabrication

Intelligent data formats such as ODB++ and IPC-2581 enable efficient data exchange. These types of data formats facilitate intelligent communication from engineering to manufacturing, minimizing errors and providing a continuous feedback loop of information from manufacturing back to design.



Library and design data management

Libraries are the foundation of PCB design. A central library provides everyone involved with better visibility of what's going on at any given time.

Centralized environment for managing PCB design data

A fully integrated PCB design data management infrastructure, from initial system definition through the design process, to release, to manufacturing. Improve individual or team design with enhanced security, consistency, metric collection and process control.

Increased overall team productivity

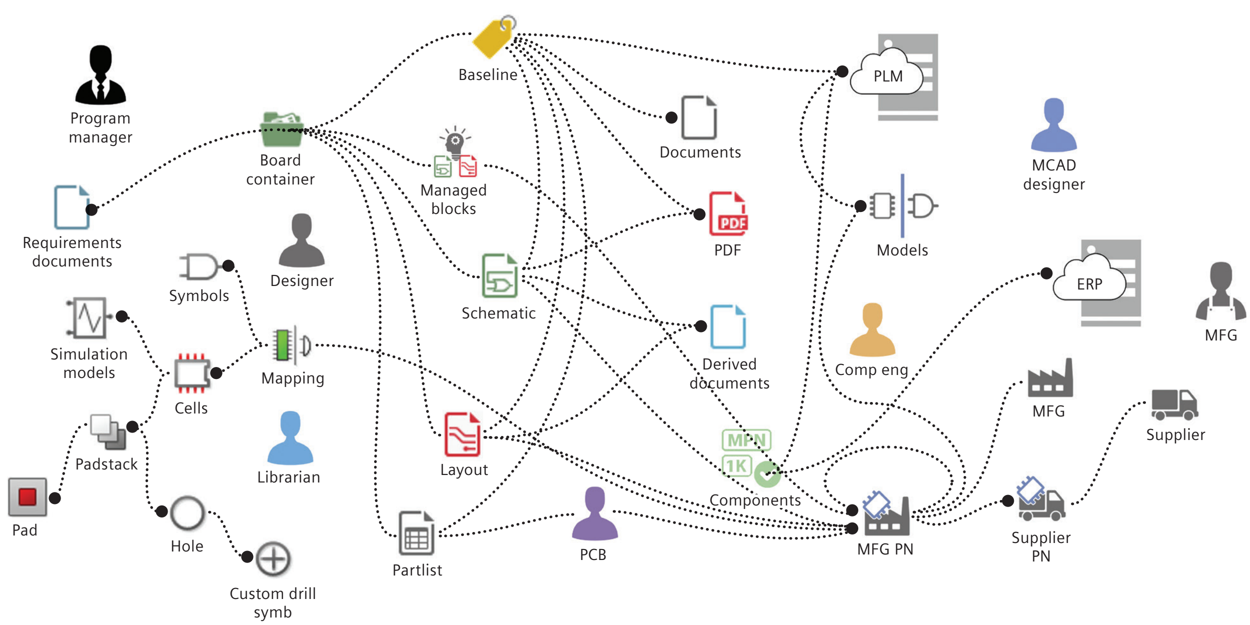
Controlled access for project team members to both design and design status information. Collaboration management for concurrent design, design reviews, release and IP management.

Reuse existing IP in the form of components and certified circuits for new products

Reusing proven elements from previous approved designs through a library of certified circuits allows new product development teams to leverage past successes for the benefit of their latest generation of products.

Direct access to comprehensive part intelligence

Integrated technical parametric information, alternates, and real-time component sourcing intelligence for parts at the point of design, provides actionable insights at every stage of the design process and synchronization with part request workflows and PLM systems.



Secure remote collaboration

Teams spread over different geographic locations need a platform for effective collaboration – without compromising data security.

Efficient design reviews

An integrated collaboration panel centralizes all processes and manages the status of design review feedback by assigning different statuses to communicate progress. It allows internal and external stakeholders such as design teams and third-party design services, to share and view joint data.

Robust project notifications

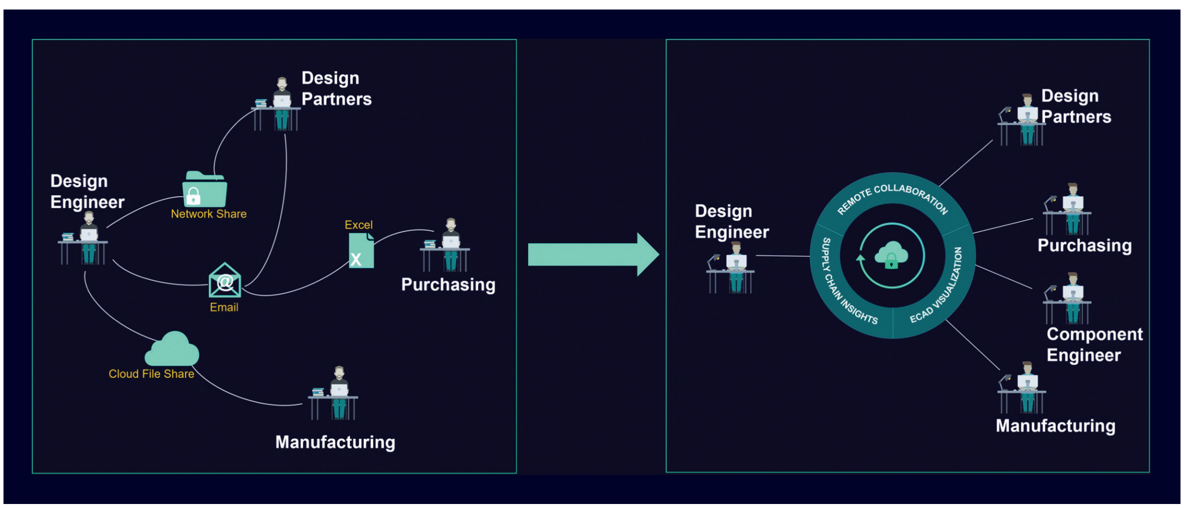
Adding comments to collaboration data provides greater context. Teams can stay aligned by using notifications for comments, by tool-integrated email or through a centralized notification panel, which helps keep the whole team up to date without the hassle of checking emails from outside the integrated tool platform or waiting for physical review meetings.

Bill-of-materials view

Including bill-of-materials (BOM) data in the collaboration platforms improves visibility for all stakeholders. Moreover, integrating with a component search engine helps with early identification of component sourcing issues.

Security-driven collaboration process

Cloud-hosted applications require rigorous enforcement of international security standards. In addition, a meticulous security process which requires admins to assign users with a license and role as well as access to specific projects, ensures controlled, secured collaboration.



Engineering productivity and efficiency

Optimize engineering productivity & efficiency to reduce design cycle time, reduce cost and lower risks.

Design automation: streamlining tasks throughout the PCB design workflow

Automating repetitive tasks leads to faster execution, fewer errors, and increased overall efficiency. It also frees up designers' time to explore various design alternatives or configurations.

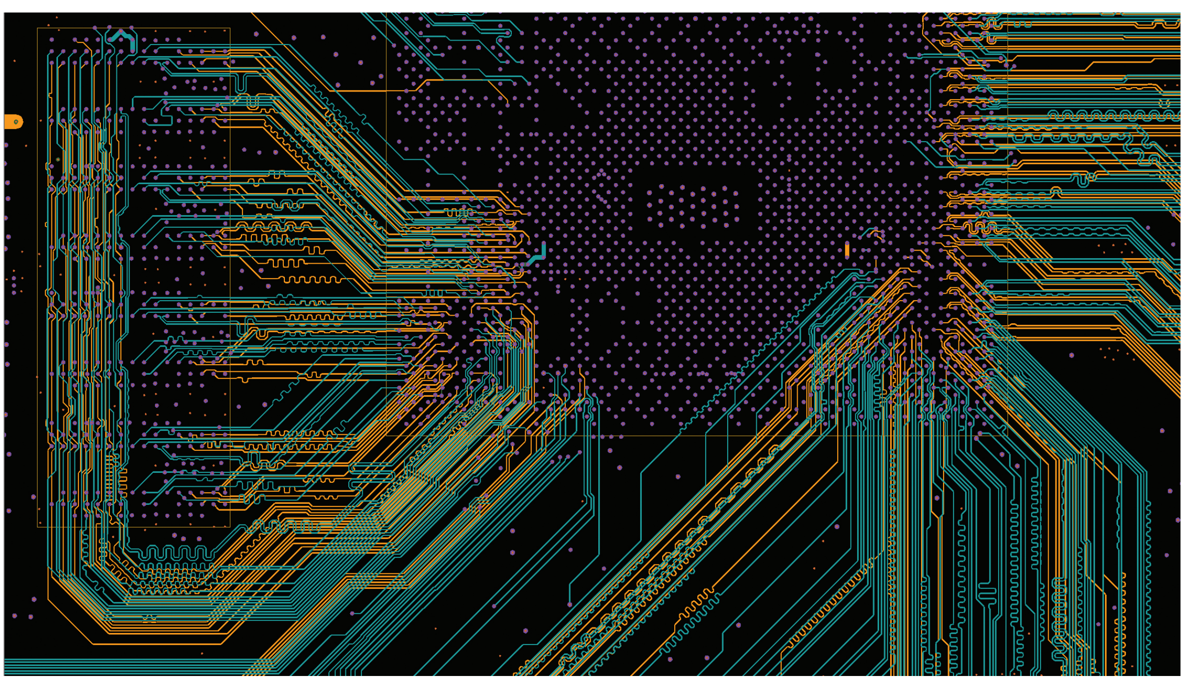
Connectivity planning and PCB routing
Automatically unravel complicated net paths and pin escapes to minimize net lengths and quickly and easily identify the best routing strategies.

Auto-assisted interactive sketch routing
Sketch routing provides integrated automated PCB routing functionalities with intuitive user control, high quality, and exceptional performance. "What-if" scenarios can be explored and changes absorbed more easily.

Hierarchical placement planning and management
Quickly visualize and implement the engineer's design intent to produce an optimal component placement. Easily address PCB real-estate assessment, also know as an "Area Study".

True parametric 3D
True parametric 3D mechanical kernel, with 3D constraints, dynamic collision detection, and batch verification. Dynamic validation of electro-mechanical designs delivers error-free results.

Mechanical component import
Easily import all physical constraints and mechanical components such as chassis or heatsinks, along with critical items and components along with their locations. Or, for true multi-board capability, import sub-assemblies from other PCB designs.



Analog/Digital/RF co-design

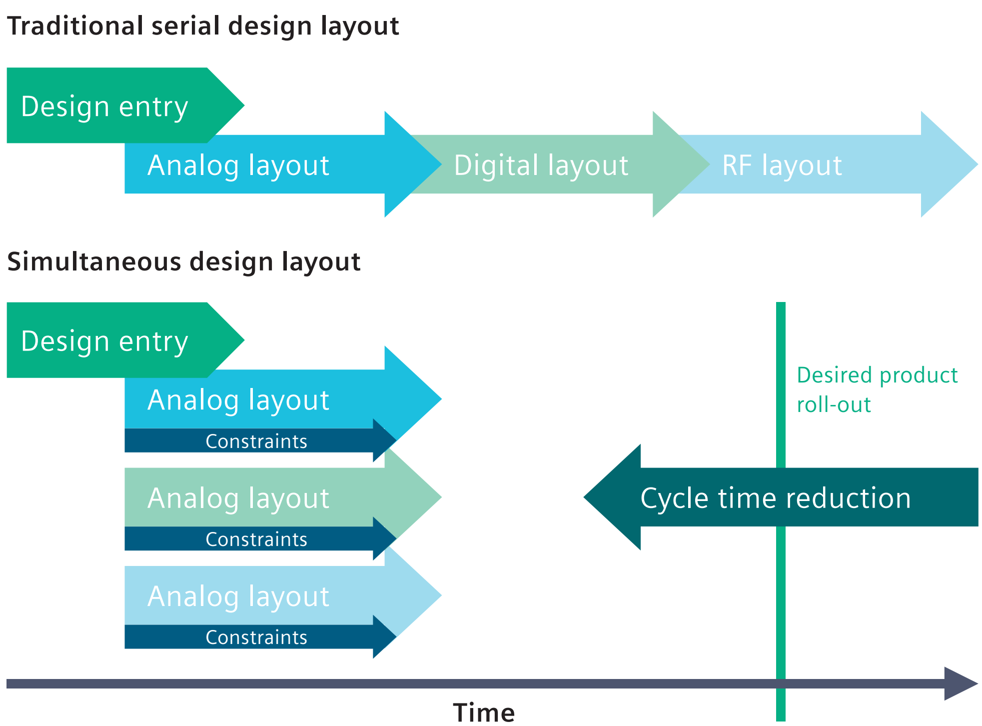
Design RF, analog and digital circuits on the same PCB – eliminating the traditional "black box" approach to RF design and siloed work.

Streamline RF tool integration
Eliminate manual data transfer with dynamic integration between the PCB design tool and industry RF tools. The library is synchronized with the circuit-simulation model counterpart in the RF simulation environment to ensure that their behavior is identical.

Accelerate design time
The ability to edit RF elements within PCB layout, combined with efficient RF tool integration eliminates workarounds and increases engineering productivity.

Optimize RF circuits in context
Modify parametric RF elements within PCB layout to optimize layout space efficiency and easily incorporate "what-if" scenarios.

Reuse verified RF circuits in other designs
With robust design reuse capabilities, teams can manage and share known-good circuits that are already approved and released, eliminating redundant design efforts.



Concurrent design

Instead of waiting for one user to finish their work and hand it to off to another in serial fashion, teams can carry out tasks in parallel and simultaneously, increasing optimization and team collaboration.

Schematic design
Multiple engineers can simultaneously design the same schematic. All edits appear dynamically to all concurrent users.

Constraint management
Work concurrently with other users who are editing the same constraint set, whether on the front-end or back-end.

PCB layout
Teams of layout designers can edit the same printed circuit board and work in a collaborative manner. No need to partition and reassemble.

Concurrent verification
Virtually prototype a design in-process with concurrent verification of performance and manufacturability.

System design concurrency
Define a multi-board system and start defining individual associated boards while the system definition is still in process.

Simultaneous review and collaboration
Control the editing mode for design reviews so that edits are restricted to one engineer while others can view, maintaining security that only one designer has active control of the design.

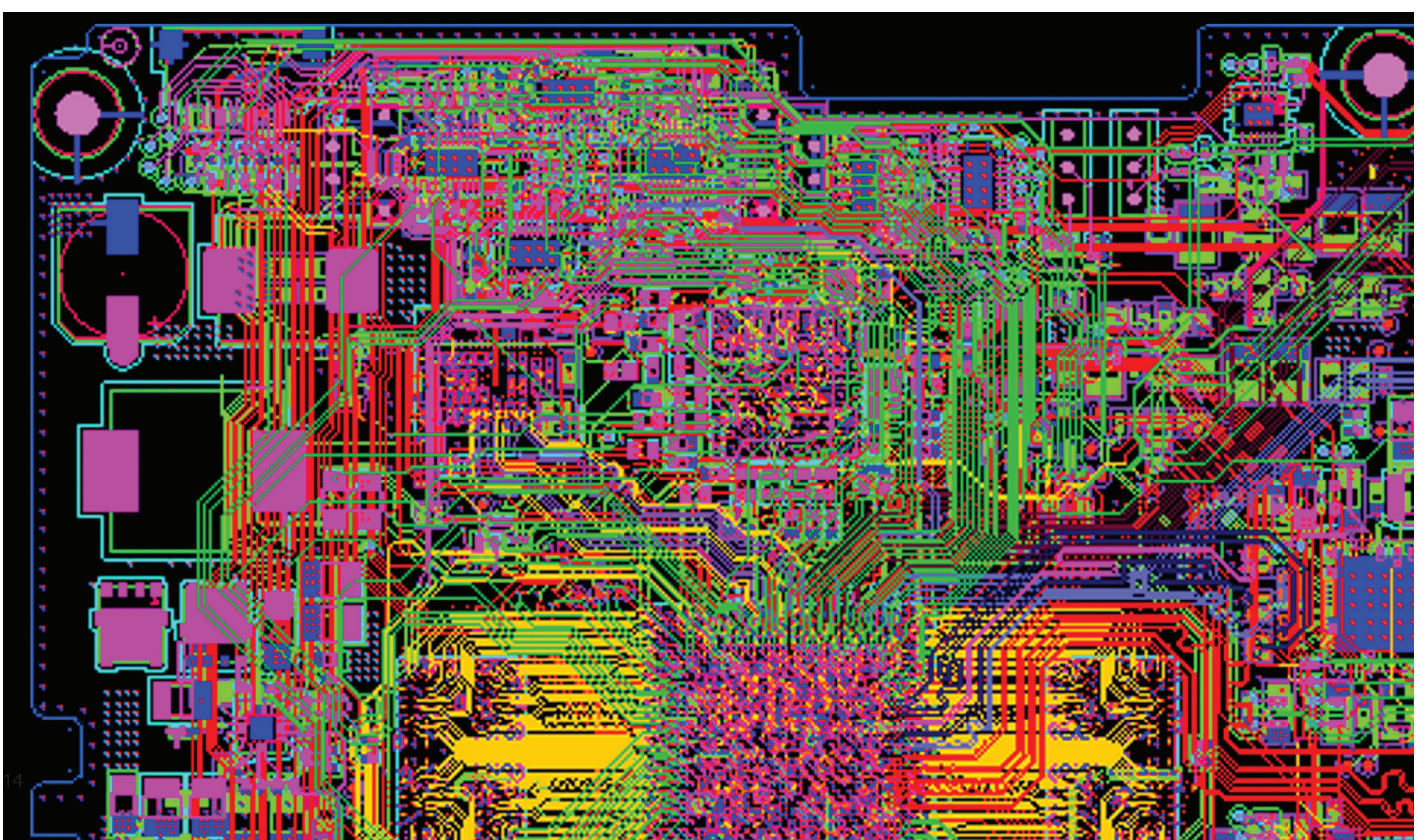
Concurrent PCB design Vs. Serial Approach

Legacy Process (and not simultaneously):

1 individual x 8hr serial working shift = **8hrs of effort**

Concurrent Design (parallel and simultaneously):

5 individuals x 8hr working shift = **40hrs of effort**



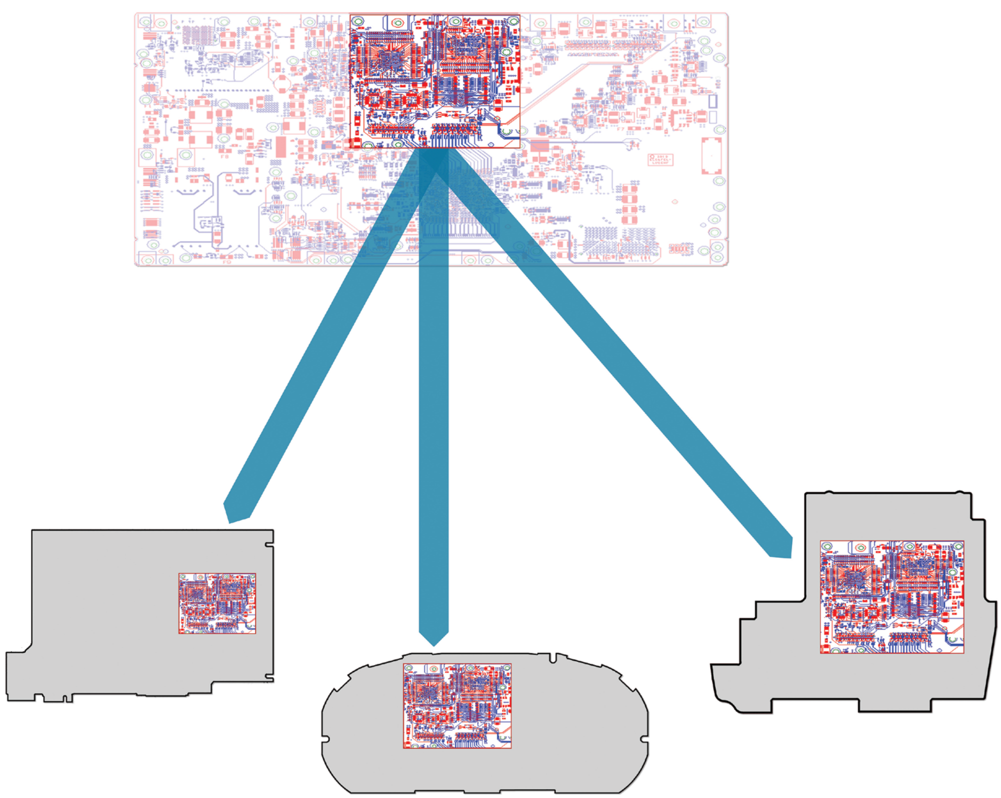
Design reuse

Utilize existing known good circuitry and proven layouts to design whole systems a lot faster, eliminate repeated efforts and minimize project risks.

Reduce cost and improve PCB design quality
Once circuits and their respective layouts are validated, approved and released, they can then be saved in the master library as critical intellectual property (IP) blocks. Then, these IP blocks can be reused over and over again in new PCB design projects. This reduces engineering efforts (costs) and improves the quality of future PCB designs.

Increase collaboration and efficiency
Foster concurrent development of today's complex PCB designs. Concurrent design methodology is proven to be faster and more efficient, especially when it come to multidomain and multidiscipline integration and collaboration.

Reuse proven logical and physical elements
Design teams can quickly build new products by publishing reusable logical and physical IP directly and then sharing this IP with others. Design reuse not only improves design cycles time, but also increases the potential for initial design testing/start-up to work exactly as expected the first time.



Constraint-driven design

Design with rules up front to help ensure your PCB design is correct by construction. By creating design constraint templates and implementing them, you increase your potential for consistent success.

Tool automation in layout

Taking advantage of automation features such as creating/editing constraints, net/constraint class assignments, signal routing and trace tuning and DRC checking within today's tools has the potential to be a game changer when implementing a constraint-driven design approach to PCB design.

Easy constraint entry

Constraint entry for a design in an easy-to-use spreadsheet-like GUI, which allows bidirectional cross-probing between schematic and layout applications.

Built-in automation

Simplify manual tasks such as identi-fying and assigning differential pairs by using automation.

Constraint re-use

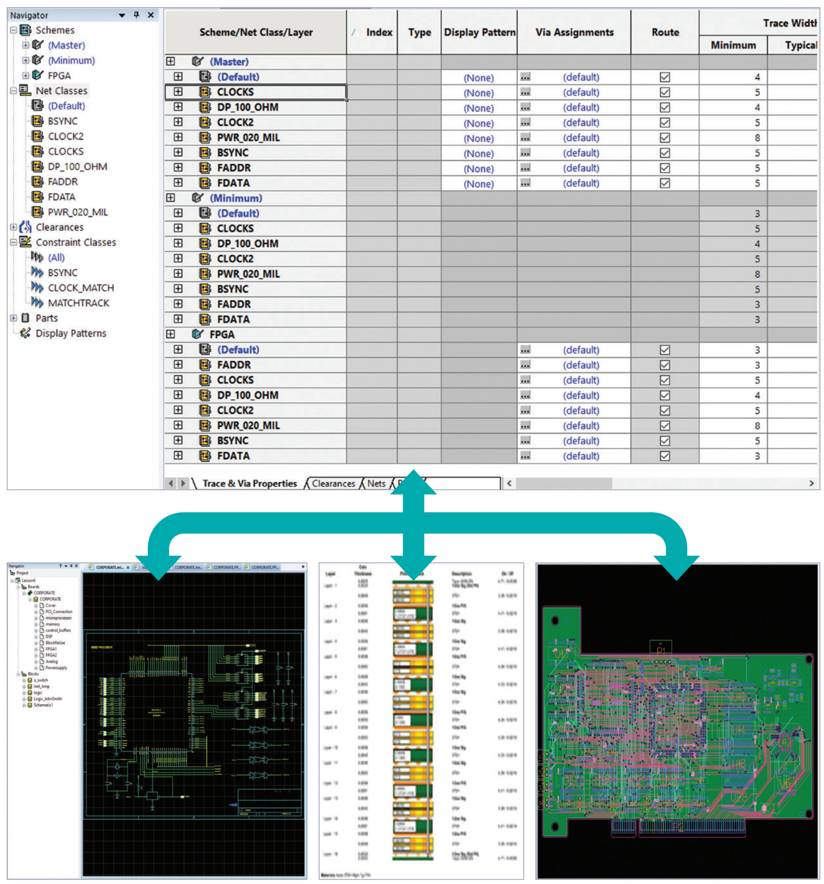
Import customizable constraint templates and modify with whatever new design constraint or requirements are necessary for the new design.

Design reviews and validation

Utilizing constraints allows easier design reviews. Prove design intent meets design requirements and customer satisfaction by ensuring validation for quality of the end-product.

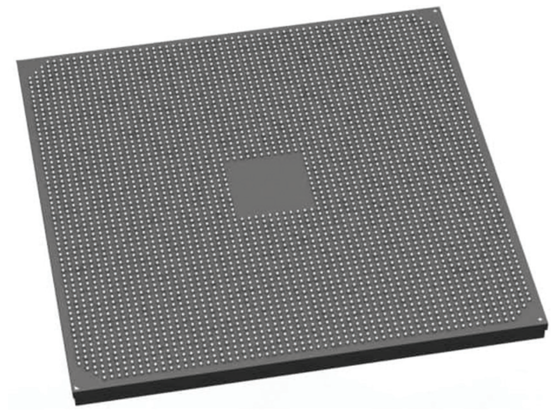
Target lengths dialog

Designers can make sure that when tuning nets, they stay within the parameters, such as minimum and maximum length and tolerance.

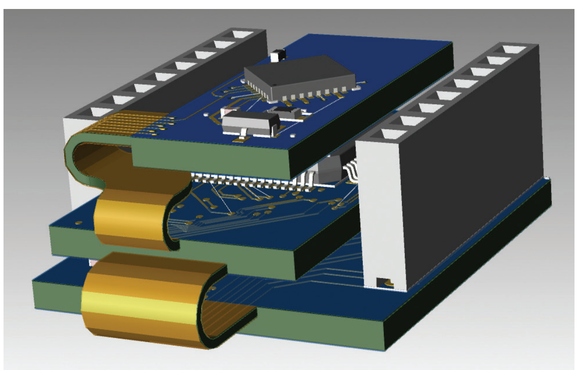


Design for advanced manufacturing technologies

The utilization of advanced manufacturing technologies can get you to market faster.

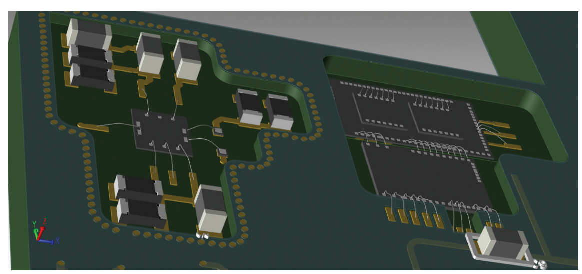


Rigid-flex PCB design
Ensure first-pass success with a design process that factors in the unique challenges of rigid-flex PCB design. This includes unique stack-up styles, PCB bend-areas, regions rules such as vias, trace and plane types, and design visualization – all in an integrated 2D/3D environment.



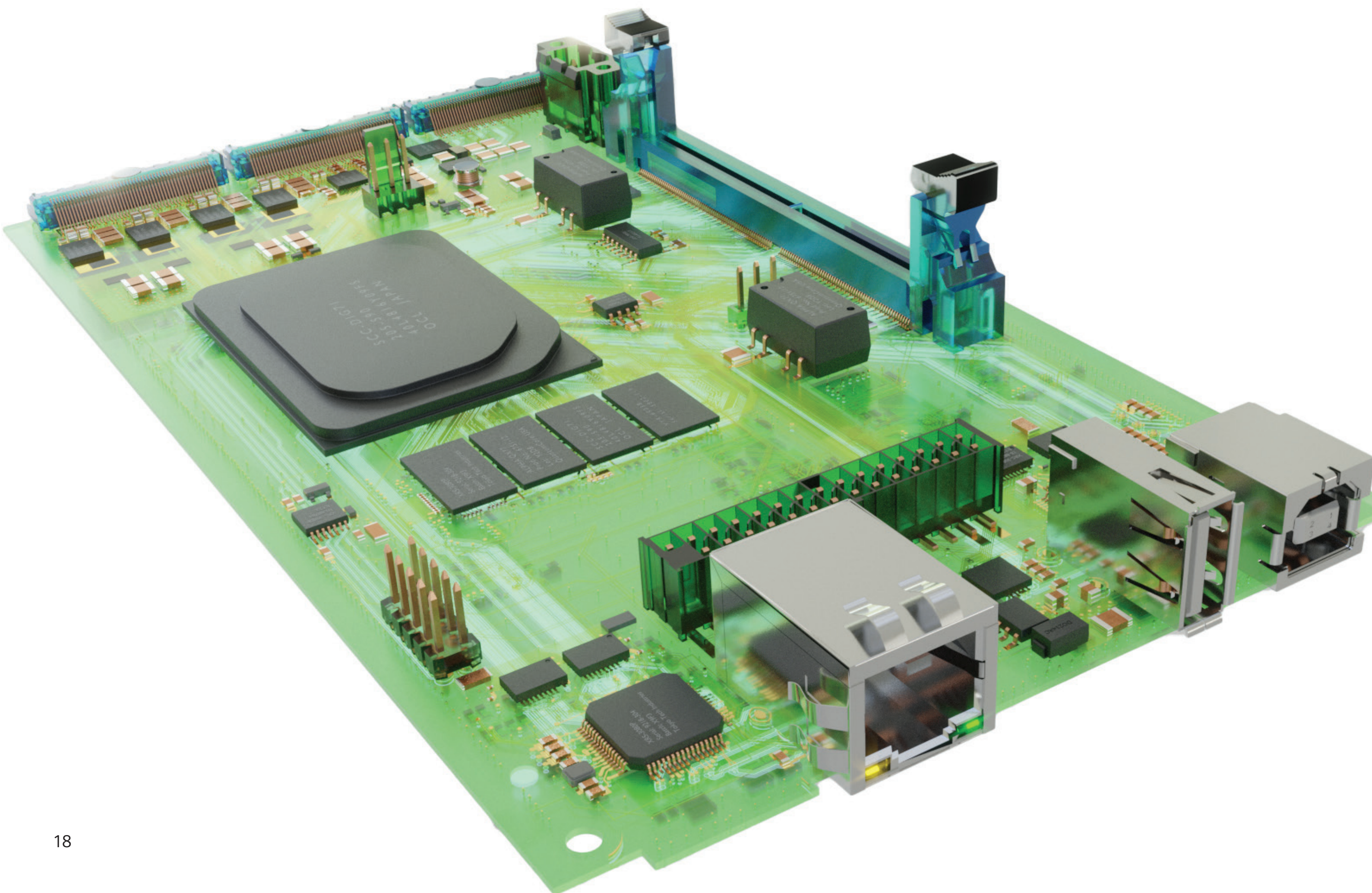
High-Density Interconnect (HDI)
Many I/Os in a small area makes it nearly impossible to route to the inner balls with normal PCB fabrication technology. What is required for these connections are layers of High-Density Interconnect (HDI) with implementation of multiple advanced via structures such as via-pad, blind vias, buried vias, and microvias.

Embedded actives
Tackle heat spreading and stray inductance issues with design embedding. Active and/or passive components are placed in cavities inside the PCB, embedded with the PCB stackup.



Digital prototype driven verification

Shift-left design methodology integrates design verification tools throughout the PCB design process to let designers find and fix errors where they happen, instead of waiting until later.



Digital-prototype driven verification

A shift left approach to schematic analysis, thermal analysis, signal integrity, power integrity, and manufacturability.

Schematic analysis
Fully inspect all circuitry and nets on a schematic using an extensive, intelligent model component library and perform schematic parametric analysis in parallel with design capture; more than simple connection checks.

Signal integrity analysis
Explore signal integrity issues early in the design cycle to make key design decisions when options are easier to compare and implement.

Thermal analysis
Use 3D computational fluid dynamics (CFD) software to predict airflow and heat transfer in and around ICs, packages, boards and electronic systems.

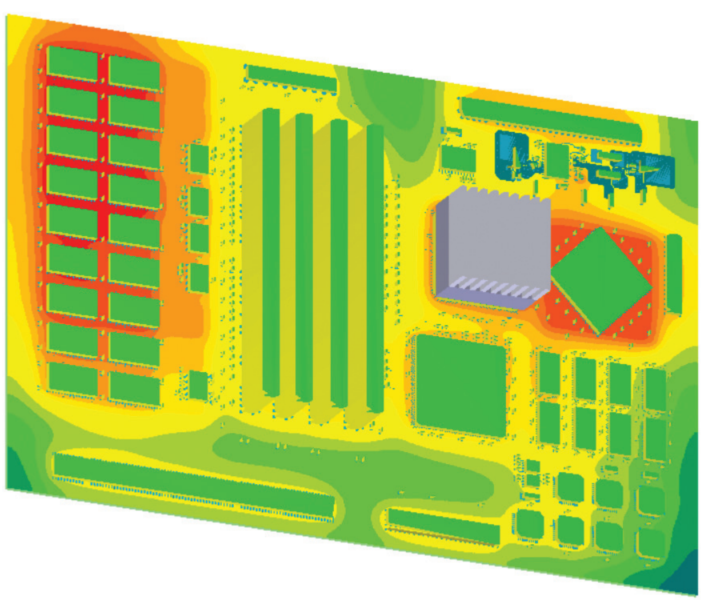
Power integrity analysis
Accurately model power distribution networks and noise propagation mechanisms, during both pre- and post-layout phases of the PCB design process.

Analog mixed-signal analysis (AMS)
AMS expands standard SPICE-based circuit analysis to leverage industry advances in PCB modeling and simulation, accelerating circuit development and speeding verification.

Manufacturability
Run a set of automated checks to validate a design against fabricator and assembler capabilities and execute concurrent DFM checks throughout release-to-manufacturing.

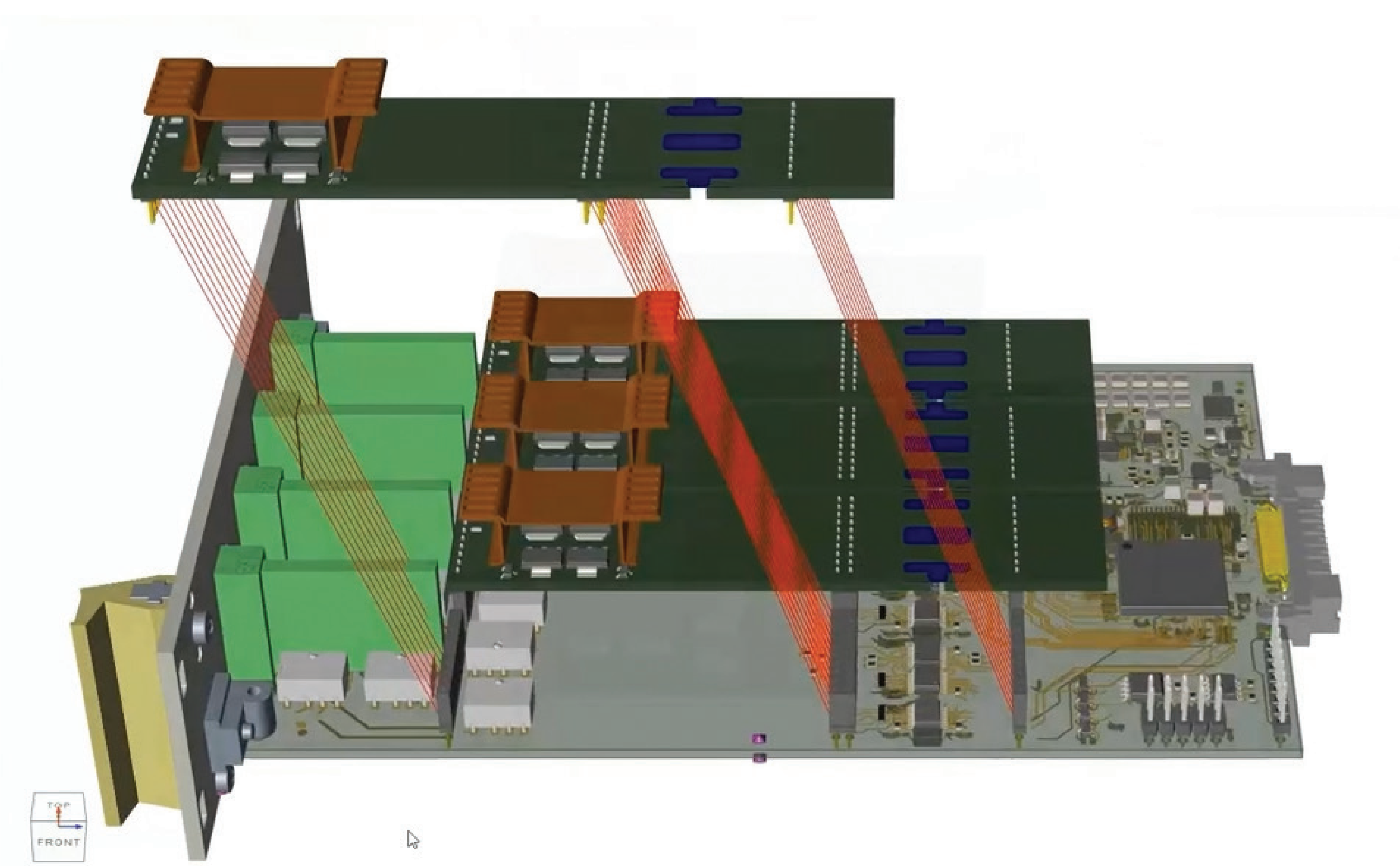
Test analysis
Monitor testability considerations and calculate test point requirements, as well as testability coverage.

Electrical rule checks
Identify design violations that lead to signal integrity, power integrity and EMI/EMC issues, eliminating manual inspection and bottlenecks.



Model-based systems engineering

A digital twin allows teams to perform simulations early, enabling trade-off analysis and optimized system integration.



Multi-board systems design

Replace inefficient paper and manual processes with an automated, fully integrated, collaborative workflow for electronic multi-board system definition.

Systems design automation
Project architects can define and capture the hardware description at the logical system level down to the logical/PCB level, including the logical definition of wires, cables and backplanes.

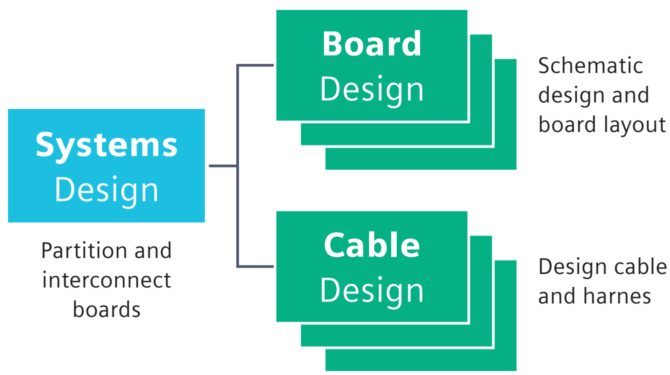
Built-in automation
Visualize and implement the system engineer's design intent, to create optimal function location and interconnectivity across the system.

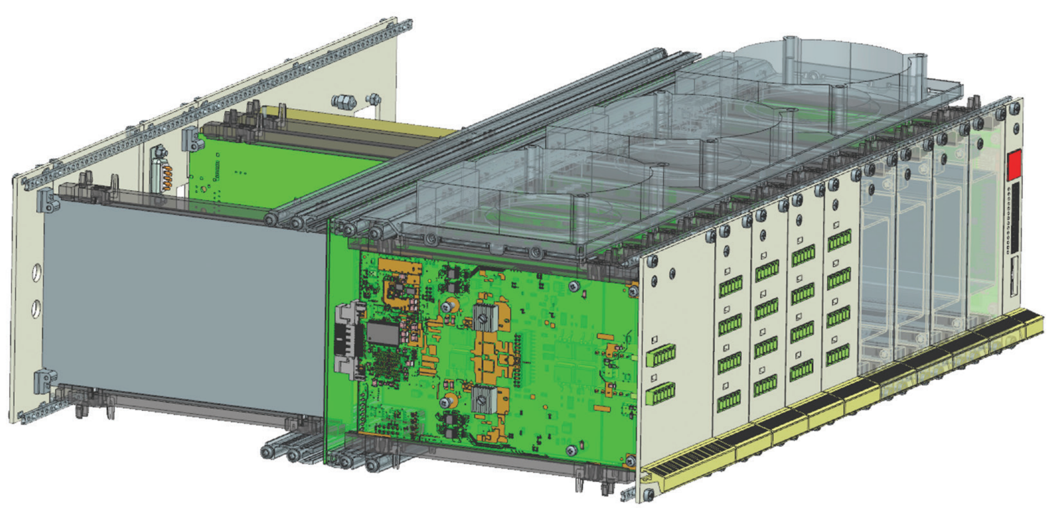
System modeling
Virtually prototype multi-board systems for mechatronic behavior, signal/power/thermal performance, and manufacturability.

System design reuse
Keep system design elements and their associated data intact and where designers can easily access it, even in a dynamically changing environment.

Cable design
Definition and optimize cable connectivity, with MCAD integration and management through to BOM and manufacturing drawings.

Collaborative concurrent design
All-domain collaboration using versioned, unified data with notification, cross-probing, mark-up, and comments for consistent, integrated design process management.





Electrical/electronic co-design

Circuit boards usually interface with other circuit boards via connectors, cables or harnesses. A scalable digital thread facilitates collaboration between these disciplines.

Connector management

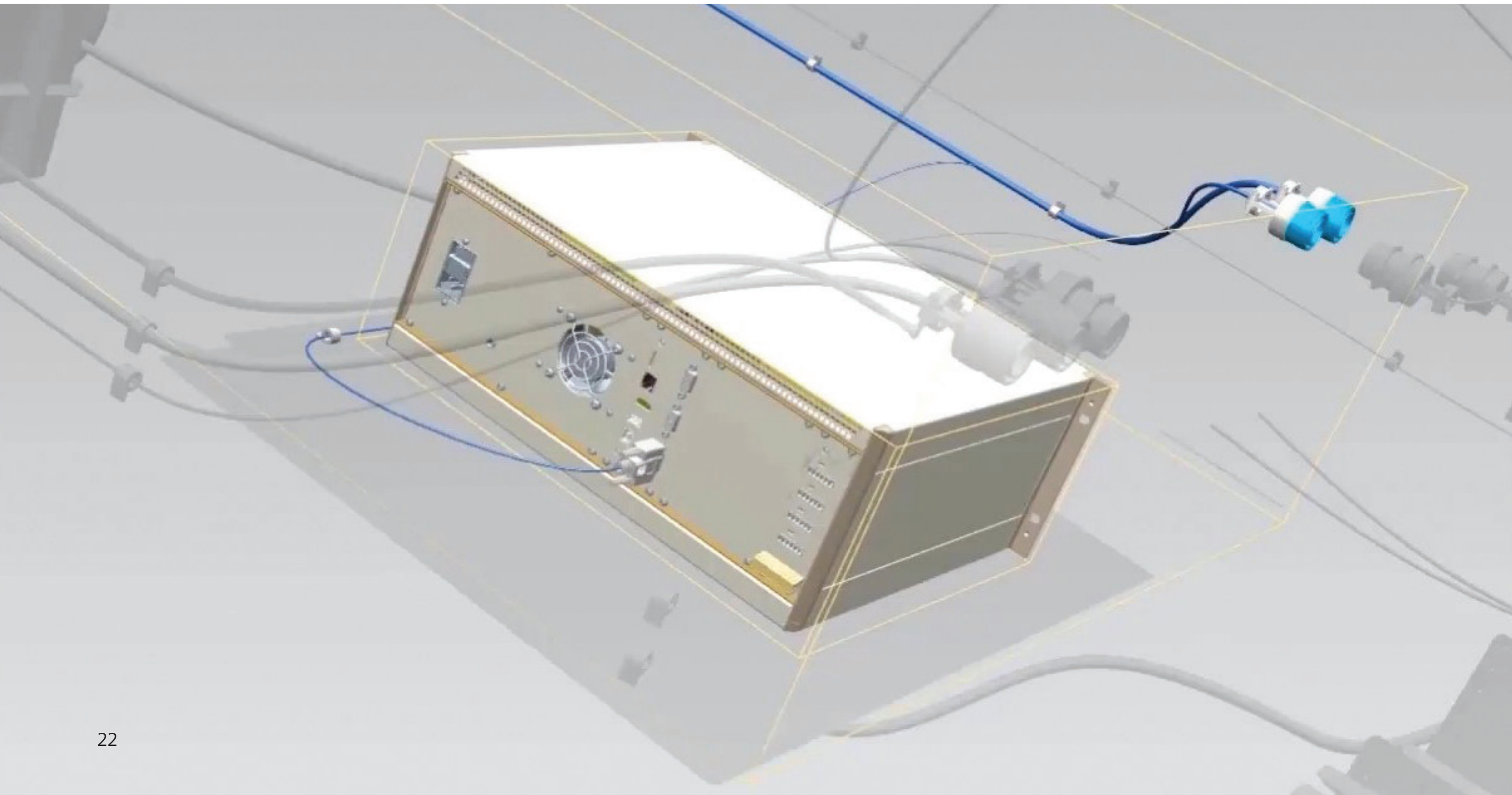
Add, manipulate and update connectors with on-the-fly creation of parameterized connectors with connector expansion and pin number modification by using a simple stretch command. With a high degree of automation, designers can create and modify connectivity using generic connectors without concerning themselves with the actual physical details of the connector.

Multi PCB and cable integration

Synchronize the content of logical boards and associated PCB schematics using a bi-directional process. As the PCB schematic is synchronized with associated logical boards, the PCB schematic contains connectors and system level blocks that are, in effect, hierarchical blocks. Therefore, board designers can push into these blocks and define logic on the underlying schematic to realize each specific function of the system.

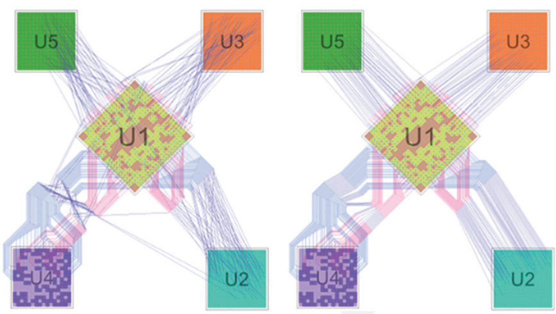
Integrated cable design

Automate selection of parts by automatically adding wires, multicores, terminals, tapes, and all other cable components for ready-to-manufacture cable designs, including bill-of-materials and manufacturing drawings. Automatic calculation of quantities, such as 'true' manufacturing wire lengths, ensures correct-by-construction cables.



FPGA/PCB optimization

Powerful and extremely high pin-count FPGAs provide engineers with significant opportunities for increased features and functionality while reducing the cost of their products. But with increased complexity comes significant challenges in integrating these devices onto the PCB.



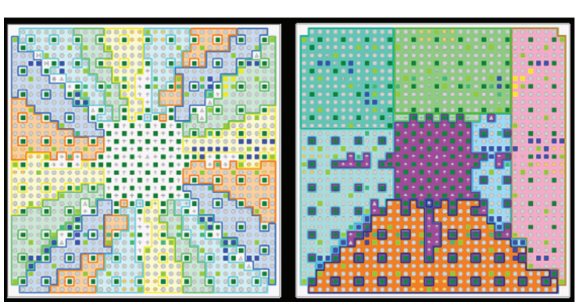
Connectivity needs to be optimized across multiple devices.

Flow integration
Keeping schematic, PCB layout and FPGA databases in sync allows users to better control the project's design data flow. This can be managed automatically through a synchronization assistant.

Signal grouping
Manage the signal complexity of high pin count FPGAs and accelerate the pin assignment process by creating signal groups. Each group may define the interfaces of particular hardware functions to be implemented later on in the FPGA.

Pin partitioning
Pin partitioning allows for better PCB connection planning, customized symbol generation and minimizing the pin swap rules within the group, leading to better control of pin optimization and improved net unravelling.

Signal and pin assignments
Auto assignment, supervision of signal standards, assignment by simple drag and drop, support for operations on sets of objects and dynamic filtering all make signal-pin assignment a simple operation.



Ten banks partitioned into seven custom pins groups.

Automated part and symbol generation
A set of powerful features makes symbol creation easy, fast and error free, while still allowing full control of the symbol creation process. Compared to manual symbol creation, time is reduced from hours or days to minutes.

Floor planning
Floor planning can be done before and during the PCB layout process. Designers have the clear advantage when they can make FPGA pin assignment changes right from the project's early stages.

Net line unravelling
Automatic optimization of signal-pin assignments while respecting pin specific rules and constraints enables highly efficient net line unravelling.

FPGA multi-instance
The same FPGA device will often have different logical functions shared across projects or even shared within a single project. An I/O optimizer can fully support these situations automatically during project development.

Multi-FPGA optimization
Using an optimization algorithm, evaluate all possible connection combinations to arrive at the optimum interconnect. This minimizes net cross-overs arising from the initial assignment and enables higher route completion rates.

IC/Package/PCB co-design

There are three primary considerations when assimilating design data from multiple IC, package substrate, and PCB design domains into a single environment:

Integrating and assembling design data from the three design domains
Visualizing multiple design domains together as a system within a single design tool is key to an ideal co-design process. This can be done by generating and maintaining an abstract model of each design domain.

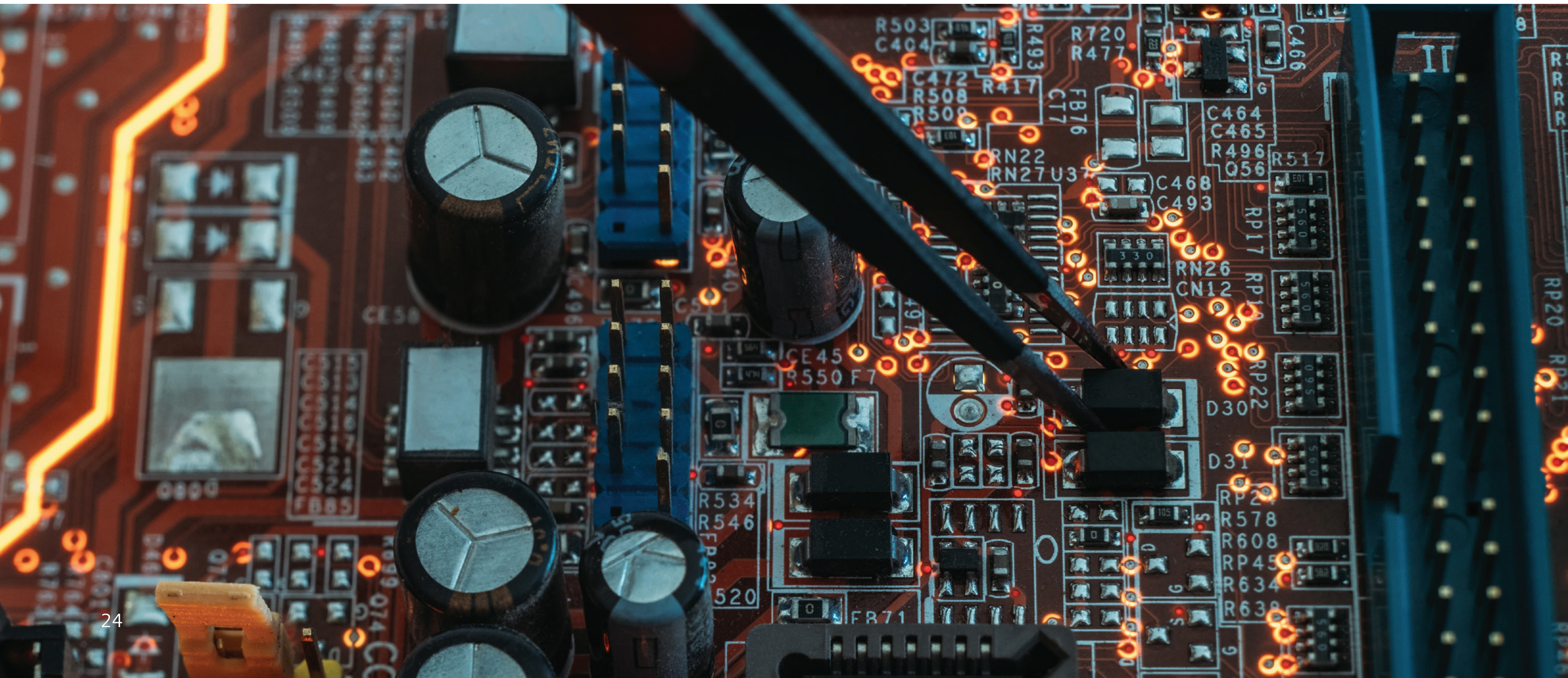
It is important to consider the level of layout detail required: the abstract model for PCB and package substrate designs should include detail of their trace routing and vias, internal component placement and wire-bond elements and definition of the internal logical connectivity.

A holistic approach to connectivity management
As signals or power rails transition from die-to-package and from package-to-PCB, the logical net names may change or even be split into more than one connection. Connectivity management must support logical combining, splitting, aliasing, and pin mapping of signals as they transition from design domain to design domain.

A tremendous amount of flexibility is required to support the various traditional methods of capturing connectivity within each domain so that each domain expert can work in the mode most familiar to them or most convenient for a specific task.

Handling incremental changes occurring within each domain
When changes are made within the abstract model they must be propagated back to the original design domain. This may include changes to the connectivity, pins, changing pin pitch, or changing component locations and orientations.

A robust ECO process is required to identify and manage incoming changes through an "accept/reject" model for each design instance. Using an "accept/reject" approach facilitates the co-design process by allowing control of the timing and type of changes that are propagated to or from each design domain.



Supply chain resilience

Bringing supply chain information such as long lead times and potential shortages to the engineer's desktop can alleviate sourcing and production challenges before it's too late.

Respond effectively to challenging supply chain situations

It's time to left shift sourcing and component data to provide visibility to long lead times and potential shortages right as you are designing your PCB.

Best known part availability

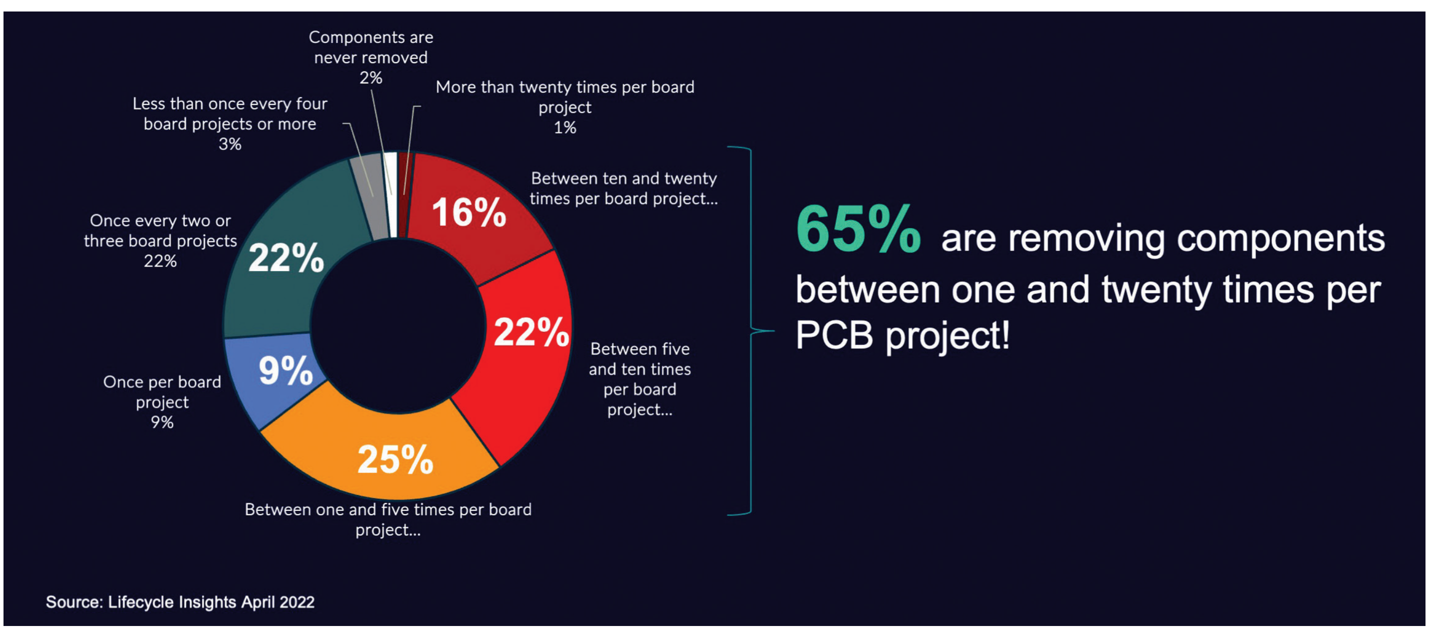
During the initial schematic creation phase of the design process, engineers typically start off by using approved best-known parts. However, supply chain disruptions often cause parts to be either unavailable or available only at an increased cost that is beyond initial targets. Real-time access to detailed intelligence for availability, price, and parametric criteria can help prevent such choices which will have to be replaced later in the process, with greater repercussions.

BOM validation

By the time initial BOM analysis and validation are completed, it is late in the design process. This causes loopbacks. Compounded by inefficient collaboration, sourcing validation takes days to weeks before part issues are identified and before feedback is received by engineering. The best practice would be dynamic analysis of BOM "health" for requirements, pricing, availability, suppliers, and risk mapped to the part master list.

Validation of alternates

Validation of alternates is a daunting manual task for engineering because there is no formal way to compare alternates, which means that validations take more engineering effort. This can be eased by an online search through a direct connection to the content provider of your choosing. When you find a suitable alternative part or set of parts, it would map directly to the supplied data model with no additional intervention.



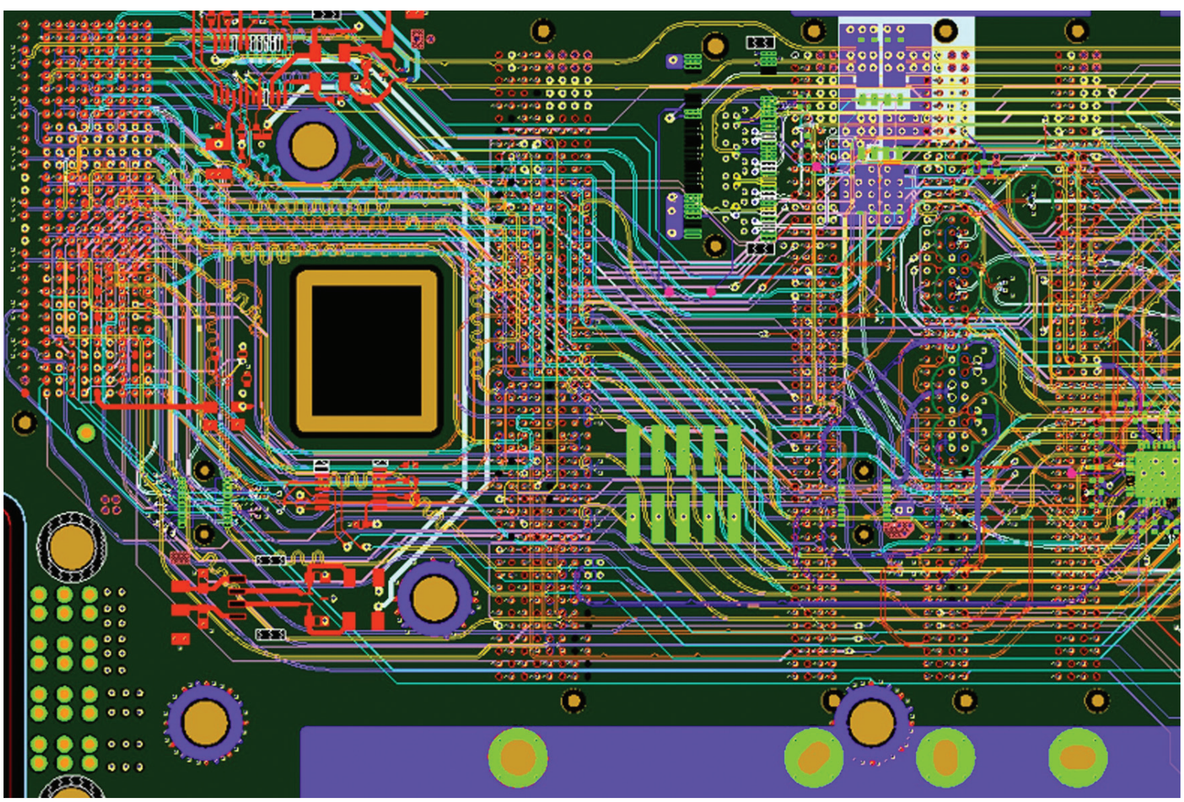
Conclusion

The complexities of modern PCB design necessitate a comprehensive approach that integrates various aspects of the design process. The five pillars of digital transformation for electronics systems design - digitally integrated & optimized, engineering productivity & efficiency, digital prototype-driven verification, system-level model-based engineering, and supply-chain resilience - provide a roadmap for companies to navigate these complexities.

By implementing these best practices, companies can optimize their processes, improve team collaboration, reduce design time, lower engineering costs, and increase the reliability and quality of designs. These practices also enable teams to handle the increasing product, organizational, process, and supply chain complexities in the electronics systems design ecosystem.

The future of PCB design lies in harnessing the power of digital transformation, offered by the Siemens Xcelerator product portfolio, including Xpedition, HyperLynx, Valor and PADS Professional. By doing so, companies can stay ahead of the curve, delivering innovative products that meet customer expectations and drive the industry forward.

Embracing these best practices in PCB design is not just about surviving in a complex electronics landscape; it's about thriving and leading the way in technological innovation. The journey of PCB design is ongoing, and as we look to the future, these five pillars will continue to guide us towards success.



Siemens Digital Industries Software helps organizations of all sizes digitally transform using software, hardware and services from the Siemens Xcelerator business platform. Siemens' software and the comprehensive digital twin enable companies to optimize their design, engineering and manufacturing processes to turn today's ideas into the sustainable products of the future. From chips to entire systems, from product to process, across all industries, Siemens Digital Industries Software – Accelerating transformation.

Americas: 1 800 498 5351

EMEA: 00 800 70002222

Asia-Pacific: 001 800 03061910

For additional numbers, click here.

© 2024 Siemens. A list of relevant Siemens trademarks can be found here.
Other trademarks belong to their respective owners.
85974-D7 6/24 H