PCB DESIGN: IMPROVING PROFITABILITY WITH PCB DESIGN BEST PRACTICES

November, 2015

→ **Tracy Woo,** Research Analyst,
Product Innovation & Engineering (PIE)

→ **Reid Paquin,** Research Analyst,
Manufacturing, Product Innovation & Engineering (PIE)

Report Highlights

p2

The electronics industry is rich with competition. Product differentiation through performance is key to being an industry leader.

p5

Best-in-Class companies were found to consistently outperform their peers on product launch, cost, quality, and revenue targets.

p6

The Best-in-Class implement a PCB design process that optimizes system design, design data management, collaboration, PCB virtual prototyping, and DFM.

p12

Best-in-Class companies are more likely than their peers to implement technology tools that can span across the PCB development process.

Based on the experiences of over 175 respondents, this report will examine how successful companies have optimized their Printed Circuit Board (PCB) design process. Specifically, how Best-in-Class companies implement best practices that address system design, design data management, collaboration, virtual prototyping, and design for manufacturability.

ABERDEEN GROUP

A successful PCB design starts with a process that optimizes system design, design data management, collaboration, PCB simulation, and design for manufacturability (DFM).

Differentiating Products

Respondents were asked to select what is most important to differentiate products with PCB assemblies (all respondents):

Performance / Capabilities – 50%

Cost – 17%

Accelerated time to market – 13%

Form factor / Size – 7%

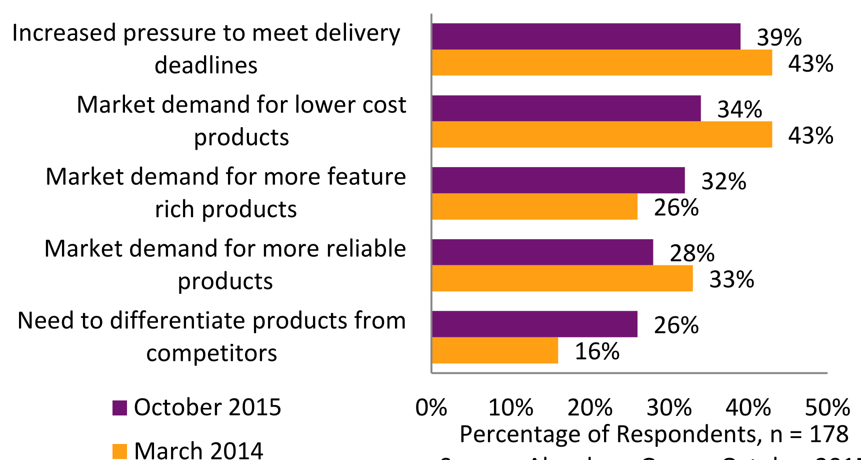
Regulatory compliance – 7%

Energy efficiency – 4%

Competing in a Challenging Environment

As more items incorporate electronics every year, the need to efficiently design PCBs to meet those demands increases. All companies today struggle with the pressure of meeting their new product development goals but, for the those within the electronics industry, this is especially true. As shown in past Aberdeen research on PCB Design, cost and speed to market are still the top drivers causing companies to invest in improving their PCB design process. However, over the past 18 months, the secondary pressures have started to shift across the industry (Figure 1).

Figure 1: Top Pressures Driving PCB Design Improvements

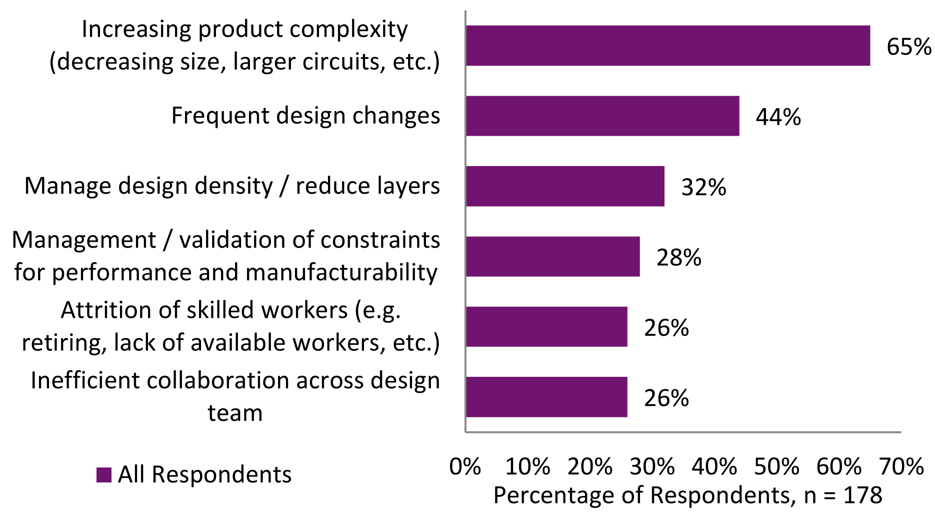


Source: Aberdeen Group, October 2015

As the data shows, the need to introduce differentiated, feature-rich products has increased a fair amount from March 2014. Product differentiation through performance, in such a competitive environment, is key to being an industry leader, a separation that is made in its electronics (see sidebar). A company that fails to distinguish itself by being first to market or unique in functionality will find itself bypassed by those who can. Creating high performance products without compromising time to market and cost are the main focus of companies that fabricate and design PCBs.

Unfortunately, PCB design and fabrication are usually limiting factors in getting a product to market, at the root of the issue, rising product complexity. In fact, 65% of respondents cited increasing product complexity as their top challenges for PCB design (Figure 2).

Figure 2: Top Challenges in PCB Design



Source: Aberdeen Group, October 2015

The large part of the issue is the customer's demand for smaller packages but higher functionality. This dichotomy between decreasing surface area and increasing circuitry is where the problem lies. In addition, changes occur frequently throughout the development process which, if not properly managed, result in delays and increases in cost. Another significant internal challenge on PCB companies is demographics - the engineering workforce is limited and creeping towards retirement age. The combination of these challenges raises an important question: how do companies meet customer expectations while also hitting schedule deadlines, budgets, and quality standards? The

"We are trying to overcome the fact that we are a small company, with a small budget, trying to design products that are comparable in technology used and capabilities to companies with much larger budgets. We find that we often have to forgo the use of some technologies because we can't afford to design, prototype, then risk having to redesign them again."

~ Director of Product Development, Consumer Electronics Manufacturer

Aberdeen's PACE Methodology

Aberdeen applies a methodology to benchmark research that evaluates business Pressures, Actions, Capabilities, and Enablers (PACE) that indicate corporate behavior in specific business processes:

Pressures – the external forces that impact an organization's market position, competitiveness, or business operations.

Actions – the strategic approaches that an organization takes in response to industry pressures.

Capabilities – the business process competencies (process, organization, performance, and knowledge management) required to execute corporate strategy.

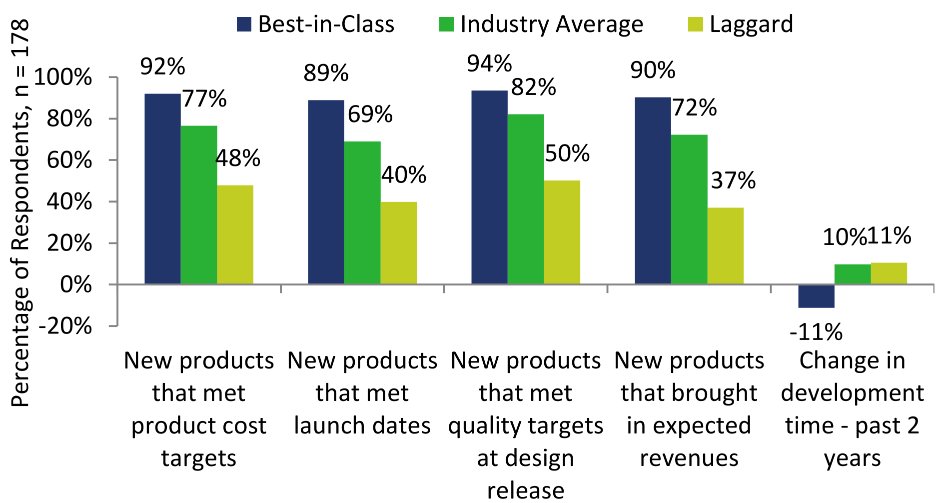
Enablers – the key functionality of technology solutions required to support the organization's enabling business practices.

good news is that there are companies who have figured out a solution.

Defining the Best-in-Class

To define Best-in-Class companies, Aberdeen used five metrics that measure an organization's ability to deliver their new products successfully. Aberdeen categorized participants as Best-in-Class (top 20% of performers), Industry Average (middle 50%), or Laggard (bottom 30%); we also refer to a fourth category, All Others (Industry Average and Laggard combined). Figure 3 highlights the performance of the three maturity groups.

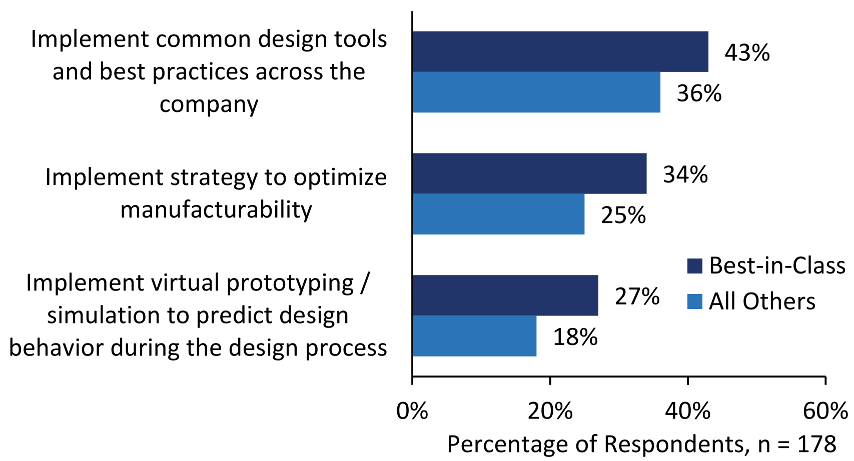
Figure 3: Metrics Used to Define Best-in-Class



Source: Aberdeen Group, October 2015

Best-in-Class companies consistently outperform their peers in all five metrics selected. Most notably, they are the only class to see a *decrease* in development time over the last two years, which is the top pressure all companies currently deal with. In cost, quality, and expected revenue over 90% of their products met these targets. The Best-in-Class are in this league for a reason. They have a clear understanding of how to optimize the PCB design process (Figure 4).

Figure 4: Top Actions Moving Forward



Source: Aberdeen Group, October 2015

In order to maintain the balancing act between schedule, budget, quality, and functionality, companies that are Best-in-Class focus on optimizing five areas in their PCB design process:

- System design
- Design data management
- Design collaboration
- Virtual Prototyping
- Design for Manufacturability (DFM)

Focusing on these five areas of PCB design help define and optimize the design process. This improved process also leads to greater consistency within Best-in-Class designs, allowing them to meet scheduled launch dates. This has also allowed them to do a better job of avoiding data integrity issues that hurt quality, lead to waste, and drive up costs. Their peers cannot say the same, which shows in their poorer overall performance.

"Our main design challenge is manufacturability and functionality. To address it, we have conduct pre-design meeting with R&D and production to capture any potential issue to understand which part of the design is critical and need to take care of. This process is in progress and should be able to reduce some design issues."

~ Product Development Manager, Industrial Equipment Manufacturer

"We have enacted a PCB Design process flow with design checks. These added checks have lessened errors and improved the quality of the first iteration of boards, saving time and money."

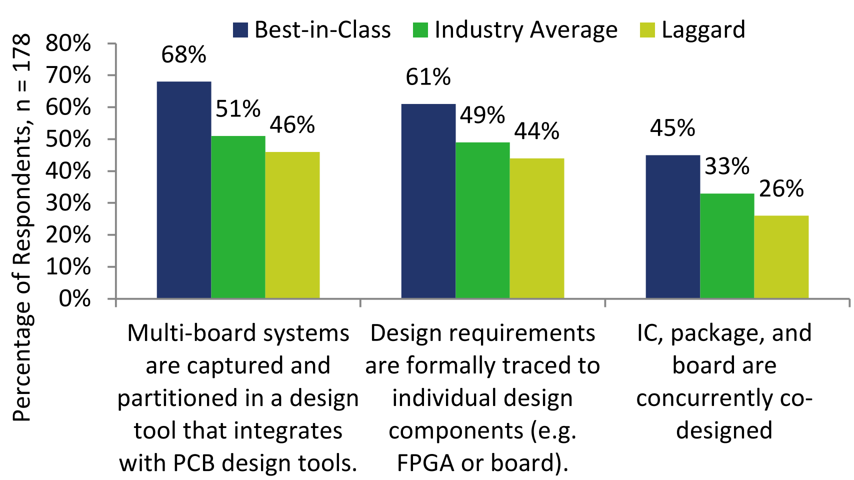
~ Product Development Manager, Telecommunications Equipment Manufacturer

System Design

The inclusion of system design is essential for any multi-board product, especially when hardware functionality spans several PCBs. Traditionally, antiquated tools, such as Microsoft Visio or Excel, have been used to conceptualize the product, describing each board functionality and their interconnects. The open platform of these products is alluring. However, the downfall is the manual entry of data, particularly when rising product complexity runs board connections into the thousands, if not tens of thousands. Even with a dedicated role to this assignment, changes throughout the development process from design to ECO will clash with an inflexible system.

What's needed is a system design tool that facilitates defining and partitioning electronic systems from the logical abstract level to the PCB. Best-in-Class companies are 33% more likely than the industry average to use such a software enabler. In addition, they are more likley to formally trace requirements to individual design components when compared to their peers (Figure 5).

Figure 5: Best-in-Class Focus on System Design Tools



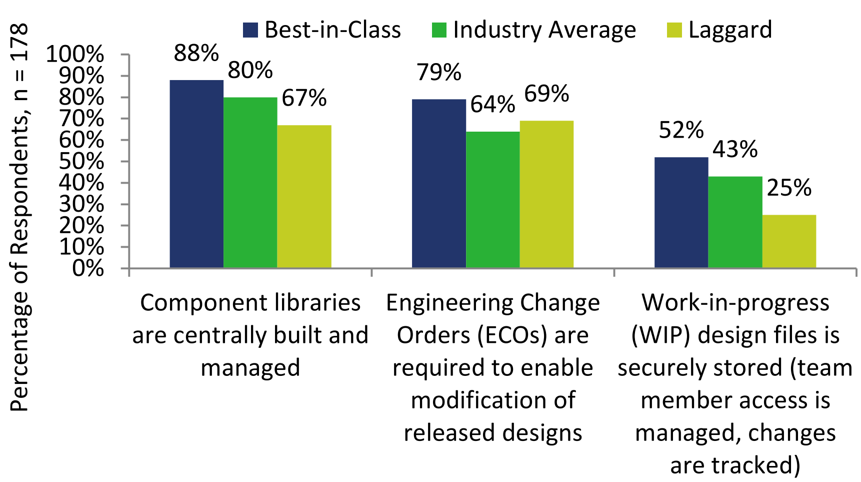
Source: Aberdeen Group, October 2015

Using a single environment for multi-board applications, that partitions electronic systems down to their functional blocks, reduces the need for manual data handling, data duplication, and data transfer errors. In essence, a system design tool provides value-added efficiency and productivity.

Design Data Management

An area that most designers would like to ignore but can't, is design data management. Droves of data are generated during the design process. Given its sheer size and complexity, managing the data is not an easy task. Designers are already overloaded without taking on the very painful task of handling reams of data. PCB data is very dynamic – having concurrent design processes means the design evolves as different people work on it. An often used, but ineffective solution, is putting all data on a network drive and hoping for the best. The problem is this path does not ensure the much-needed consistency required in product development. To alleviate this step, Best-in-Class companies are more likely to use a single repository to store all relevant information (Figure 6).

Figure 6: How Are You Managing Your Design Data?



Source: Aberdeen Group, October 2015

Fast Fact

Designs are released to enterprise systems (ERP, PLM, etc.) for production using a consistent and automated process:

Best-in-Class – 61%
Industry Average – 44%
Laggard – 35%

"A common challenge is that components have been built by multiple librarians to differing standards. We chose a common standard, IPC Nominal land patterns, and now we are slowly replacing all components with the new standard land patterns. The result has been less questions from the assembly houses and a faster turnaround."

~ PCB engineer, Telecommunications Equipment Manufacturer

Throughout a project, design data will change and evolve. A company should know what and how that change has occurred. The following key steps should be used to aid in that process:

- As stated previously, store all data in a single repository. The latest version should be available without duplicates.
- Along that same vein, create a universal library for the whole company. Prevent time wasted in creating duplicate parts or footprints. Re-use parts when possible, there's no need to re-invent the wheel.
- Have the managers control how data is stored and who has access. This ensures continuity throughout the project.
- Be prepared for the event of a disaster and have a plan.
- Use a process that tracks the release of approved designs and their status afterwards.

For Best-in-Class, automation is also key (see sidebar). During the design release, Best-in-Class companies automate the process of design release to PLM and ERP systems in order to smoothly transition the design from development to production. Accurate and consistent management of that data is crucial to the long term organization of a company's future. Those who optimize their organizational process of design data management will be well positioned for success.

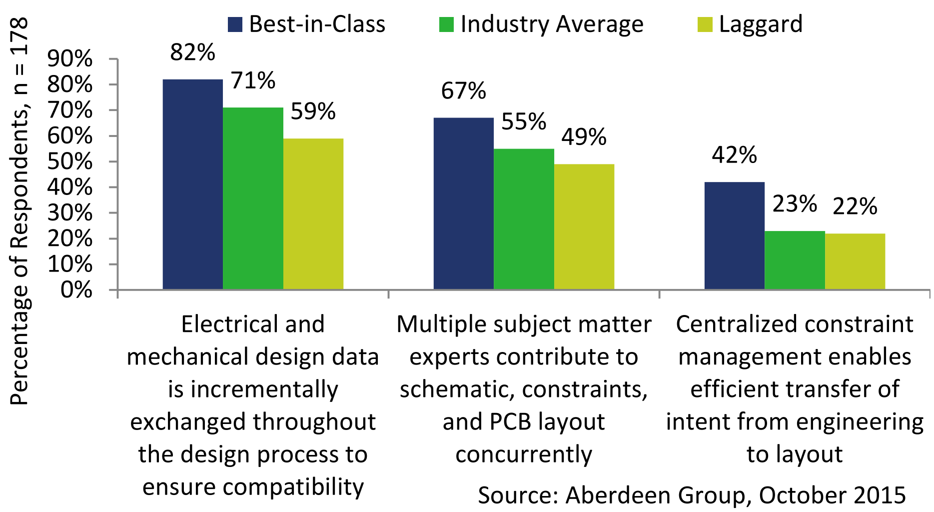
Design Collaboration

Besides organizing design data, the biggest challenge in today's advanced products is breaking free from the tradition of designing in isolation. PCBs are now tightly weaved into the mechanical and software parts of the design. Shrinking boards, increased pin counts, clock signals getting faster, and board layers growing are the result of the wave of "smaller-faster-denser". Collaboration can be a major hurdle both within a discipline (e.g. concurrent schematic, layout) as well as between disciplines (e.g. ECAD/MCAD,FPGA/PCB, etc.). The design process needs to happen in incremental exchanges throughout the design process to ensure compatibility. Best-in-Class companies are 39% more likely than the Laggards to design through incremental exchanges (Figure 7).

"By introducing concurrent PCB layout tools we have deployed up to three people on the same design concurrently to achieve project deadlines."

~ PCB engineer, Aerospace & Defense Company

Figure 7: How Are You Collaborating on Design?



Source: Aberdeen Group, October 2015

Projects designed in series, where one group follows the next, lengthen development schedules because one part is held in limbo until the other is finished. Groups working in isolation often have a difficult time integrating designs. Companies are becoming increasingly distributed and rely upon global design teams, as well as processes that aid collaboration, which are vital to any project's success. As a result, the design of a powerful product requires a development process that is tightly aligned across all engineering domains to ensure compatibility.

"The first action the company undertook, was to create a central PCB design department. This was the first step towards harmonizing the design methodology and design rules. This also enabled multiple designers on one job. The throughput was the major improvement."

~ PCB engineer, Large Computer Equipment Manufacturer

Another important aspect of collaboration is the pooling of resources both in people hours and expertise. Because of this, it is no surprise to see that Best-in-Class companies are more likely than their peers to use multiple experts for concurrent design. Having designers from many backgrounds, both electrical and mechanical, will ensure mechanical interference or signal integrity won't be an issue in a layout. These sorts of errors are important to avoid in order to maintain high-quality while staying on schedule and under budget. To avoid these mistakes, Best-in-Class companies are 83% more likely to use a centralized constraint management process to ensure all parties are designing with the correct specifications.

To remain competitive, designers from all disciplines must adjust to considering the overall product rather than a specific domain. Having an automated process that facilitates such collaboration is essential to reach Best-in-Class success.

The Importance of Simulation

Best-in-Class companies take their performance to the next level by implementing technology enablers to support the PCB design process

Analog, RF, and / or digital simulation are used to predict design behavior:

- Best-in-Class: 74%
- Industry Average: 52%
- Laggard: 43%

Signal Integrity analysis is conducted:

- Best-in-Class: 68%
- Industry Average: 63%
- Laggard: 54%

Power integrity analysis is conducted:

- Best-in-Class: 50%
- Industry Average: 42%
- Laggard: 43%

Virtual Prototyping

The need to minimize re-spins and develop higher-quality products at faster rates has caused many companies to turn to virtual prototyping. A combination of modeling and simulation, virtual prototyping has become a common part of product design, one that Best-in-Class see the benefit of (see sidebar). PCB simulation collects key inputs from the electrical or physical design process, and tests these features against assumptions and constraints made for the overall design. This process can be performed at all stages of development from concept to schematic to layout. Simulations can range from transmission line level analog simulations to power distribution simulations. During this stage, development and optimization of constraints are performed to account for all known variations affecting design performance.This process allows for designers to correct a design early in the design cycle to ensure the board is properly constructed, the earlier an error is corrected the less cost to the business. In terms of construction, signal integrity is one of the highest concerns for PCB designers. The signal must travel from driver to receiver on time, and with recognizable quality. For accuracy, the entire transmission path must be modeled, from IC to package to PCB, including the power distribution network.

Simulation is highly-recommended to perform at a Best-in-Class level, however the data shows that this technology is not currently ubiquitous in the indsutry. This can be attributed to multiple reasons, however, a main one is the fact that simulation has typically been viewed as a specialist area. Deploying simulation tools at the engineering stage allows mainstream usage, helps to eliminate bottlenecks, and allows simulation experts to focus their efforts on the most complex issue. An example of where simulation could be helpful, saya board is designed where the input signal to a DRAM controller is ringing back and crossing the threshold again. This is problematic since re-crossing the threshold would flip the bit and most likely cause the design to fail. The solution: change the transmission line impedance from 50 Ohm to 35 Ohm, adjustthe input termination and controller impedance, as well as slow the output configuration for the controller to greatly reduced the ringing. These modifications would have been impossible to do under manual prototyping. Furthermore, finding the cause of an issue, such as ringing, would have been very difficult with just measurments alone, essentially the same as finding a needle in a haystack. Therefore, potentially hours of bench time could have been lost tracking down the issue. Where as this scenario could be easily solved through simulation and virtual prototyping..

Simulation provides the benefit of consistent and immediate feedback. The use of modeling and simulation technologies help engineers refine designs and detect potential problems without having to rework expensive and time consuming prototypes.

Simulation of the input signal into the DRAM controller

The trailing edge of the signal (blue) is spiking up over the threshold line (light green)

After circuit adjustements, the trailing edge of the signal is no longer ringing over threshold

"Yield predictors based simply on the complexity (component count, layers, size, etc.) did not effectively capture the true cost of manufacturing due to design issues. To address this issue, we have implemented DFX analysis on all incoming designs to identify and work with our customers to address prior to release and as an input into our model to ensure issues are proactively captured, addressed and resolved/understood for highest return and collaborative early supplier engagement with our customers."

~ Director of Engineering, Electronic Manufacturing Services (EMS) Company

What is DFM?

DFM refers to the designer's practice of optimizing a PCB layout to prevent board defects from occurring during fabrication and assembly. In other words, it is the practice of designing a PCB layout with the manufacturing process in mind. A DFM process can help prevent some of the most typical errors and provide for a higher board yield and more successful product launch.

DFM becomes necessary on a board in the instances when the very thin pieces of copper lines on a PCB are connected poorly to components pads or vias, he places where defects are most likely to arise. While technically the board could be correct on the schematic, small pieces of copper could detach and then re-attach to another piece of copper during assembly. This unintentional detachment could create shorts on some boards while others perform as intended. Most companies avoid these issues by ensuring manufacturing analysis occurs at multiple stages across the design cycle.

Common Terms:
DFF – Design for Fabrication

DFA – Design for Assembly

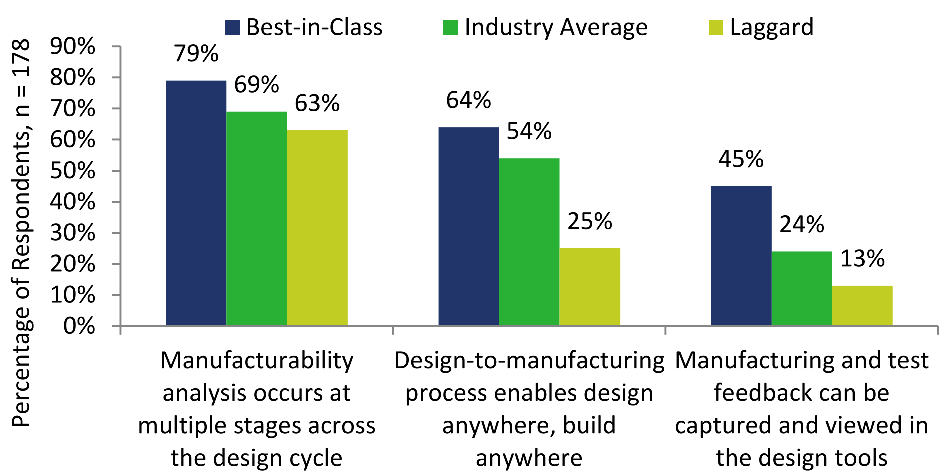
DFT – Design for Test

Companies should look to use simulation in order to lower costs and improve predictability for different scenarios.

Design for Manufacturability (DFM)

Along the same vein as simulation, DFM is an important step to take when considering the board design. Designs that require multiple re-spins are costly and will often lead to delays in the production schedule. Results from past Aberdeen research have shown that 21% of companies cited manufacturing delays from PCB errors as the driving force behind improving their data management. An often overlooked practice that can prevent these issues is a DFM process – a process which Best-in-Class companies are very familiar with (Figure 8).

Figure 8: How DFM is Being Used



Source: Aberdeen Group, October 2015

PCB layouts are intricate by nature; even the simplest boards can have design errors. Small mistakes can have huge consequences on manufacturing timelines or development budgets. A DFM process can help prevent some of the most typical errors, and provide for a higher board yield and more successful product launch. Over half of all companies perform manufacturability analysis (DFF / DFA / DFT) across the design process. Where Best-in-Class companies really distinguish themselves from their peers is by being able to capture and view feedback in their design tools. They are 88% more likely than the Industry Average to have such an ability. As a result, the Best-in-Class are better positioned to identify manufacturability problems that impact time, cost, and quality from the onset. Best-in-Class companies rely on DFM to maximize board yield and thereby meet production budgets and timelines.

Automating with the Right Tools

The Best-in-Class also use a variety of technology enablers to support PCB design. With these technologies, they are able to improve efficiency, lower product cost, and add features to competitively differentiate the product (see sidebar). The Best-in-Class use a centralized parts library to prevent duplicate work performed for the same part creation. This reduces time, increases efficiency, and ensures accuracy of the component. The Best-in-Class also use tools that support collaboration and enable them to perform concurrent design. These successful companies place emphasis on using tools that allow for multiple users to concurrently access, edit, and save PCB designs from any location. What is clear when examining technology use – it's critical to implement tools that can span across the PCB development process. With design tools specifically intended to take advantage of these areas, much of the process becomes automated, leading to Best-in-Class performance.

Key Takeaways

The truth of today's electronics market is that companies must develop products faster and more cost-effectively. With narrow profit margins and increasing functionality demands, this is not an easy task. However, optimizing the five areas of PCB design - system design, design data management, design collaboration,

Best-in-Class Utilize the Right Tools

Best-in-Class companies take their performance to the next level by implementing technology enablers to support the PCB design process

Centralized PCB parts library:

- Best-in-Class: 85%
- All Others: 75%

PCB Data Management:

- Best-in-Class: 79%
- All Others: 52%

Design for manufacturability tools:

- Best-in-Class: 66%
- All Others: 38%

Design collaboration tools:

- Best-in-Class: 55%
- All Others: 42%

Concurrent design tools:

- Best-in-Class: 45%
- All Others: 31%

simulation, and DFM - will help put a company in the same league as the Best-in-Class. To optimize these areas, companies must:

- **Find a process that captures multi-board systems and integrates them well with PCB design tools.** Using a single environment for multi-board applications that partitions electronic systems down to their functional blocks reduces the need for manual data handling, data duplication, and data transfer errors.

- **Store and control all data from a single repository. Mark any changes with strict revision control.** There's no need to re-invent the wheel. Make sure reusable, qualified blocks are accessible and used to minimize design time. During a hectic manufacture and design build, strict revision control will ensure all parties are using the most current data. Changes to the design should be communicated to all parties in one place to ensure the most accurate transfer of information.

- **Design in incremental exchanges between domains.** There's a need to provide compatibility between engineering groups during the design process. Projects designed in series, where one group follows the next, lengthen the development schedule. Groups working in isolation often have a difficult time integrating designs. Best-in-Class companies are 39% more likely than Laggards to design through incremental exchanges.

- **Perform simulation and virtual prototyping early and often throughout the design process.** Prevent costly errors in the design by performing simulation in the initial stages of development. Best-in-Class are more likely than their peers to perform analog, digital, or RF simulations.

- **Perform DFM validation during the layout process**. An often overlooked process, DFM will help keep board re-spins to a minimum. It allows the manufacturer the chance to offer helpful advice to the designer in preventing errors such as acid traps, slivers, and starved thermal pad connections. These are costly mistakes that can severely impact board yield and manufacturing costs. DFM users were 16% more likely to bring in their expected revenues than non-DFM users.

Optimizing these key areas of PCB design are crucial to stay competitive in the electronic product market. Eliminate barriers between design teams. Find ways to check designs through DFM and simulation early and often throughout the design. Most importantly, build a cohesive process that brings the entire system together. Taking these steps will allow you to bring products successfully to market.

For more information on this or other research topics, please visit www.aberdeen.com.

Related Research	
Managing PCB Design Data: Reducing Risk by Breaking Down Silos – April 2015	*PCB Design: A Guide to Optimizing Design Engineers*; March 2014
NPI Velocity in Discrete Manufacturing: The Hidden Cost of Late Products; November 2014	*The Engineering Workforce Problem: Doing More with No More* April 2013

Author: Tracy Woo, Research Analyst, Product Innovation & Engineering (PIE) (tracy.woo@aberdeen.com)
Reid Paquin, Research Analyst, Manufacturing (reid.paquin@aberdeen.com)

About Aberdeen Group

Since 1988, Aberdeen Group has published research that helps businesses worldwide improve their performance. Our analysts derive fact-based, vendor-agnostic insights from a proprietary analytical framework, which identifies Best-in-Class organizations from primary research conducted with industry practitioners. The resulting research content is used by hundreds of thousands of business professionals to drive smarter decision-making and improve business strategy. Aberdeen Group is headquartered in Boston, MA.