Technical Note

A Comparative Analysis of Computer-Aided Design Tools for Complex Power Electronics Systems

Maide Bucolo 1,2, Arturo Buscarino 1,2, Luigi Fortuna 1,2,*, Carlo Famoso 1, Mattia Frasca 1,2, Antonino Cucuccio 3, Gaetano Rasconà 3 and Giovanni Vinci 3

1 Dipartimento di Ingegneria Elettrica Elettronica e Informatica, University of Catania, 95125 Catania, Italy; maide.bucolo@unict.it (M.B.); arturo.buscarino@unict.it (A.B.); carlo.famoso@dieei.unict.it (C.F.); mattia.frasca@dieei.unict.it (M.F.)
2 IASI, Consiglio Nazionale delle Ricerche (CNR), 00185 Roma, Italy
3 STMicroelectronics, 95100 Catania, Italy; antonino.cucuccio@st.com (A.C.); gaetano.rascona@st.com (G.R.); giovanni.vinci@st.com (G.V.)
* Correspondence: luigi.fortuna@unict.it

Abstract: Companies working on semiconductors must currently assure the customers of not only the performance of the semiconductor device per se, but also its performance when it is implemented in a real board, therefore including the role of parasitic effects. It is therefore very important to evaluate, especially during the design phase, not only the single device, but the complete board and their mutual interactions. This consideration opens a new area of investigation in the field of electronic systems engineering. In the current literature, the problem of a software evaluation of parasitic dynamics and electromagnetic effects on printed boards is addressed from the point of view of researchers. Moreover, it is fundamental to have a complete view of the various tools that could be usefully adopted from the perspective of manufacturers. This is the main motivation of this technical note, which performs a comparative analysis of the most prominent software tools for printed circuit boards’ (PCBs) simulation. The main features, the key aspects, and the limitations of the software packages are analyzed in terms of the industrial design of power electronics devices, in order to ensure efficiency and fastness in the semiconductor market.

Keywords: computer-aided design; power electronics; parasitic dynamics

1. Introduction

The need for more compact, lighter, and smaller electronic systems, less sensitive to uncertainties due to construction and easier to industrialize, has always pushed towards the design in printed technology by means of printed circuit boards (PCBs).

The idea of the possibility of creating a system that would allow conducting tracks and electronic components to be combined in a single support dates back to the early 20th Century. From those pioneering tests, based on the use of various materials, relative to conductors and insulators, we arrived at circuit realizations with extremely sophisticated solutions capable of leading to multilayer PCBs, with a very high density of components, incorporating the same electronic components in the PCB matrix. This trend has recently also affected power electronics, eliminating the connection elements, such as cables and connectors, among the various switching, power, control, and filtering components [1].

When the technology for the realization of electronic systems using PCBs left the military field, where it was initially developed, to land on electronics for civil uses, it revolutionized the production of electronic devices, deeply affecting the lifecycle of the electronic product, undermining its value as a highly specific element, to the point of bringing it back to a consumer product [2].

Several common objects contain electronics, which makes them more “intelligent” and better performing. The most striking case, which has had a notable acceleration in recent
years, is that of the automotive sector. Furthermore, advances in the biomedical field show how electronics affect the human body more and more, to the point of integrating closely with it.

However, the ease of realization and the relatively low production costs linked to pushing automation, which involves the device from the components to the PCB, have reversed the relationship between construction cost and repair cost, making the producers of electronic systems lean towards the design of easily replaceable and nonrepairable electronic boards.

This change leads to reduced costs for production companies, but it increases the amount of waste material. PCBs, in fact, contain several components, but the damage to a single part of them leads to the complete substitution of the entire board. This practice contributes significantly to the e-waste problem [3].

The complexity of the projects and the ever-increasing miniaturization of the components do not allow leaving the realization of the design of a device to the human component alone [4]. The designer, right from the conception of the electrical circuit, must be supported by IT tools that allow not only dealing with the complexity of the design, but also ensuring high production performance by reducing production times and increasing operational success from the first prototypes [5].

In this context, electronic design automation (EDA), or the design of electronic systems created with the help of specific software, plays a fundamental role. PCB design is today, more than ever, at the center of the industrial activities of the most advanced countries, and the design software for systems is one of the elements that makes the difference from a competitive point of view in the continuous industrial challenge of innovation and production [6], as PCBs are now considered as complex engineering systems due to the high degree of interaction among their parts and the devices’ behavior.

In this paper, we performed a critical comparison among the most prominent computer-aided design tools for PCB design with particular attention to applications related to power devices. As a technical note, this paper is therefore intended to emphasize how advanced simulation packages could be used in order to analyze the behavior of complex boards with power electronics devices even in the design phase, prior to physically implementing the first prototypes. In [7], the lateral and vertical PCB loop inductances were extracted by using the Advanced Design System (ADS) software obtaining results that are accurate and reliable and allowing for deriving a comprehensive model of the considered device within a PCB layout, while in [8], the ANSYS software was used to predict the performance of a high-inductance voltage regulator. Instead, here, a novel topic for applied research dedicated to the semiconductor market is the focus, considering the point of view of the developer, rather than that of the researcher.

In the semiconductor market, high-level costumers require manufacturing companies to ensure the reliability of the boards where the specific device is included in order to guarantee all the electromagnetic compliance. In most cases, the design phase is the result of a continuous exchange between manufacturer and customer; therefore, this means continuously having a view of the parasitic dynamics of the board itself prior to physically realizing it. In order to do this, a continuous upgrade and feedback from the software houses providing analysis tools are mandatory.

The paper is organized as follows: In Section 2, an introduction to the workflow of PCB design is outlined. In Section 3, the resolution methods, spatial approach, and import/export capabilities are discussed. In Section 4, the software tools are analyzed and compared, and in Section 5, a case study related to a board actually under the design and test phases is discussed to show the useful information gained from computer-aided design tools. Finally, Section 5 draws some final considerations.

2. PCB Design Workflow

The main steps of the PCB design flow, necessary for the increasingly sophisticated and complex printed circuit design activity, can be summarized in the following points.
The PCB design software requires tools to create the circuit schematic and simulate it, then the PCB layout, with the related analysis tools, and then, the preparation for production.

In view of this, the main phases of the PCB design flow can be identified in the following points:

- Design of the schematic;
- PCB layout;
- PCB analysis;
- Preparation for production (manufacturability).

The schematic design is the first phase of the electronic system’s design, allowing moving from the idea and concept of the product to the actual project. It consists of drawing the electrical diagram of the product, i.e., a simplified representation of the electronic circuit using conventional symbols. In this phase, the software for the creation of the schematic must be made available to the developer, in a single working environment with all the needed tools. This environment must include tools for circuit design, simulation, component selection, library management, and signal integrity planning [9].

With the layout design [10], we enter the core of the PCB design flow, since, in this phase, the actual design of the printed circuit is defined, which can be very complex and made up of many layers.

Especially in the case of multilayer circuits, where the complexity is very high, the PCB design tool must give the developer the possibility to automate the operations, but at the same time allowing it to maintain control, in a perfect collaboration between the human mind and the software. The PCB analysis is necessary for the verification and simulation of the conditions of use of the circuit and allows the creation of its virtual prototype. The analysis does not necessarily lie in a specific phase of the PCB design flow, but must be able to be carried out at any stage. Analyzing, correcting, and verifying the most critical project requirements avoid costly prototype respins. Therefore, PCB analysis is an essential ingredient to push innovation to the maximum, reducing costs and time to market.

Among the most important analyses that must be carried out in this phase, we can mention: signal integrity, power integrity, thermal analysis, and design rule checking (DRC).

Signal integrity (SI) is a set of measurements of the quality of the electrical signal. It may happen, in fact, that as the length of the track increases and the frequency increases, the signal tends to degrade, suffering effects such as noise, distortion, and loss. Such phenomena can lead to device malfunctions. The purpose of the signal integrity analysis is to analyze and mitigate these effects. This activity involves all phases of the electronic product’s realization, from the internal connections of an integrated circuit (IC) to the package, from the PCB to the backplane, up to the connections among systems.

Power supply analysis, or power integrity (PI), allows simulating the behavior of power distribution networks (PDNs) throughout the design flow, to analyze voltage drops and to identify areas of excessive current density in the layout. In this case, the “what-if” analysis allows comparing possible design alternatives, from the point of view of power integrity.

Thermal analysis [11,12] allows simulating the impact of both the positioning of the components and the thermal routing, i.e., the design of the tracks. An effective thermal analysis identifies, even in 3D, the “hot spots” that can potentially lead to problems and defects, allowing carrying out the “what-if” analysis, i.e., the comparison between different design alternatives, both in positioning and in dimensioning cooling elements.

Design rule checking (DRC) allows checking whether the rules have been observed in the layout. This leads to the possibility of effectively analyzing the problems of electromagnetic interference (EMI) [13], electromagnetic compatibility (EMC), signal integrity, and power integrity [14]. The DRC analysis can be carried out both for a quick recognition of the networks and to directly identify some types of problems, such as anomalies in the intersections between tracks, changes of reference planes, shielding, and control of the vias.
The last step towards the creation of the PCB is given by manufacturability. In the PCB design process, it is a very important concept. This term means the ability of the printed circuit design to be produced without problems. At this stage, the developer must be able to create production documentation as integrated as possible with the design flow.

At this point, it is clear how the software for PCB design must also be integrated with the tools to manage the design for manufacturing, i.e., the analysis that allows identifying more specifically if, in every detail of the project, the PCB manufacturability requirements are respected.

The complexity of the PCD design flow makes clear that assisting the development of novel boards in the early stage of the design with suitable software packages may provide fundamental advantages to respect the market requirements in terms of reliability and production time. For this reason, leading companies in the semiconductor market need to assure their high-level customers of the high degree of promptness with respect to their requirements, continuously adjusting the design phase.

This technical note aims to provide useful insights into the choice of the most reliable computer-aided design software for this specific task. Moreover, a further fundamental requirement, considered in this technical note, is the possibility of exchanging continuous information with software developers, in order to obtain customizable solutions in a timely manner. This aspect is explicitly taken into account in the selection of the software package analyzed, thus leading us to focus on the main software houses, neglecting freeware or low-cost independent software solutions, which may meet the standards for research activities, but are lacking from the point of view of production and manufacturing.

3. Resolution Methods and Spatial Approach: Import/Export Capabilities

Whether we focus on the design of precision high-frequency or high-power electronics systems, simulation tools are critical for designing, analyzing, and validating the functionality of the circuits for their implementation on a PCB. The main reasons for performing simulations on a PCB would be to increase the likelihood of a first-pass success for the final product. Simulations can therefore be useful during the entire board layout design, aiming at different purposes in order to drive design changes and adaptations.

The commercial CAD software packages for manufacturing purposes are mainly based on three technological solutions: method of moments (MoM), finite element method (FEM), and finite-difference time-domain (FDTD) techniques.

The method of moments (MoM) is a complete, full-wave numerical technique suited to deriving the solution of open-boundary electromagnetic problems. The first book introducing the use of the MoM in electromagnetics dates back to 1993 [15]. This technique allows analyzing electromagnetic radiation, scattering, and wave propagation problems efficiently, in terms either of time or computational resources. The MoM is an integral equations solver that focuses on the integral form of the Maxwell equations; therefore, it follows an approach that is not based on their differential form, as finite element or finite difference time domain methods do [16].

The finite element method (FEM) is a numerical technique that performs the finite element analysis (FEA) of the considered spatiotemporal physical phenomenon. The FEM is based on separating the spatial domain into patches onto which it finds local solutions satisfying the differential equations, under the boundary conditions specified on each patch. The local solutions are then merged to gain the global solution [17].

The finite-difference time-domain (FDTD) method simulates electromagnetic spatiotemporal phenomena by letting the fields evolve in time. It is suited for simulating the behavior of the model over a wide range of frequencies, allowing for reliable broadband and transient analyses. The model scales linearly with the system size; therefore, it is preferable for the electrical modeling of large structures with complex geometries. It fits also with the parallel processing paradigm. As for the FEM, the FDTD method is a true 3D field solver, which can be used to analyze arbitrary 3D structures. Not requiring matrix inversion operations for its implementation and the error sources in FDTD calculation being well
known, the method permits accurate modeling for a very large variety of electromagnetic wave interaction problems [18].

Accurate electronic circuit and package designs are required for high-performance radio frequency (RF) and ultra-wideband (in the order of GHz) interconnections, to understand and minimize undesired electromagnetic wave phenomena or to identify critical points within the PCB.

For the most generic simulation capability, full-3D solvers can be used. These tools are capable of simulating any 3D structure and solving for complex coupling, including antenna radiation patterns [19].

For PCB-related simulation, most of the features are planar in nature, and the tools can be optimized for such structures. These planar simulators are generically called 2.5D solvers.

The difference in the use of the two types of simulators is substantial, both in terms of computing resources and the time needed to complete the simulation.

In the case of full-3D solvers, several tens of gigabytes of RAM may be required, and there is a need to deploy the calculation on multiple computers in parallel, noting a calculation time of several hours. For a 2.5D simulator, the computational problems are less severe. A few gigabytes of RAM, the possibility of calculation on a single computer, and a calculation time that can reach a few tens of minutes are the particular characteristics.

The main advantage of the planar solvers is therefore the reduction of the simulation time. Additionally, the execution time does not increase exponentially with the problem size as it does for 3D solvers. The main drawback of 2.5D solvers is that they approximate the coupling between layers when signals pass through vias. This approximation is something to be aware of, and sometimes, to gain confidence in the results from the 2.5D solvers, it is good practice to run a simulation using a full-3D solver as a reference.

Another important feature that must be kept in mind when using a PCB simulation tool is the capability of the software to import PCB artwork directly from CAD packages. When performing simulations of PCBs, the ease of importing a database and obtaining the first simulations to run are very important. Each of the examined tools provides several import options. These import capabilities include filters to directly import databases from Cadence, Mentor, Altium, and AutoCad. To further enhance the import flexibility, they include filters capable of importing ODB++ databases, which are quickly becoming the standard for sending production files to manufacturing, substituting Gerber files, and therefore, almost all tools can provide such an export capability [20].

4. Comparative Analysis

There are numerous simulation software packages available to support the design and deployment of power electronics systems. We focused, in this comparative analysis, on three PCB analysis and simulation packages as they are the most diffused and provide the manufacturing companies with a dedicated assistance service: ANSYS, Cadence Design Systems, and Keysight Technologies.

In general terms, these simulation tools use similar approaches. In Table 1, an overview of the main features of the considered tools is reported.

The simulation process is outlined over the following steps:

1. **Drawing the physical model**: creation of the layout geometry and definition and assignment of material properties;
2. **Electromagnetic (EM) simulation setup**: This step includes the definition of the extent of the simulation and the boundary conditions, the assignment of ports, and the setting of specific simulation options;
3. **EM simulation**: The physical model is spatially discretized upon meshes. The field/current flowing across each mesh is approximated by a local function whose coefficients and terms are automatically adjusted to fit the boundary conditions;
4. **Postprocessing**: calculation of relevant parameters, such as S-parameters and far-field radiation patterns. This step highly characterizes the different solvers and methods as
different parameters are effectively measured with respect to the considered solution. Therefore, in this step, it is possible to discriminate the more suited solution for particular applications.

Table 1. Comparison of the main features of power electronics design software for time-domain schematic simulation.

| Features                        | Keysight Tech. PathWave ADS [21] | Cadence Allegro PSpice System Designer [22] | ANSYS Electronics Desktop [23] | ANSYS SIwave [24] |
|---------------------------------|----------------------------------|---------------------------------------------|-------------------------------|------------------|
| DC/AC Analysis                  | •                                | •                                           | •                             | •                |
| Transient Analysis              | •                                | •                                           | •                             | •                |
| S-Parameter Simulation          | •                                | •                                           | •                             | •                |
| Harmonic Balance Simulation     | •                                | •                                           | •                             | •                |
| Electrothermal Simulation       | •                                | •                                           | •                             | •                |
| Monte Carlo Simulation          | •                                | •                                           | •                             | •                |
| Cosimulation                    | •                                | •                                           | •                             | •                |

In general, assessing the suitability of a given EM analysis tool is based on several considerations, which are surely the result of the direct comparison of simulation algorithms, but also include several other important factors, such as the design flow efficiency. In these terms, we must not overlook some aspects that involve the interoperability.

Therefore, it becomes essential to understand which possibilities are offered by the tools in relation to the creation, or import, of the geometric model of the PCB, as well as the possibility of exchanging information efficiently and in real time between suite tools, without resorting to complex reworking.

Ultimately, we understand how the evaluation of the tools cannot be performing keeping in mind only the method of solving the electromagnetic problem and the results that can be obtained, but it is necessary to have an overview that allows returning a profile of the system that is capable of ensuring precise results, flexibility, and relatively short working times.

In the power electronics domain, the following types of analyses are fundamental to efficiently design PCB boards:

- The power DC analysis is fundamental to assess whether, with the required power supply, the current consumption is suitable and all the devices in the circuit are working in a safe region, without exceeding their normal ratings [25];
- With signal integrity analysis ensures the circuit bandwidth, a low cutoff frequency, the gain, the roll-off, and any peak in the frequency response. In the time domain, SI analysis allows measuring the rise/fall times and slew rate, drawing the eye patterns, and determining many parameters, such as the quality factor [26];
- The S-parameters’ calculation consists of a small-signal AC simulation allowing identifying passive components and establishing the small-signal behavior of the device under specific physical conditions;
- Parasitic extraction is the computation of parasitic effects in both the designed devices and the required wiring interconnections of a power electronics circuit. The parasitic parameters of frequency-dependent resistance, inductance, capacitance, and conductance (RLCG) for electronic products are identified and well characterized;
- Electrothermal simulation at the system level is a combined simulation of the electrical and thermal aspects of the system. The circuit model calculates the power dissipation, which is used by the thermal subsystem to evaluate the temperature [27];
- Electromagnetic interference (EMI) analysis is an important investigation tool that allows identifying the electromagnetic interferences from environmental sources on
PCBs and vice versa. In the design of a PCB, it is of fundamental importance to reduce the EMI effects in order to avoid malfunctions [28,29];

- Time-domain cosimulation is the joint simulation of loosely coupled stand-alone subsimulators. A cosimulation algorithm takes care of the time synchronization and interactions across the subsimulators.

As discussed in the previous section, EM software is classified on the basis of the geometrical dimension actually considered in the calculations:

- 2D means that transverse EM waves are analyzed on two axial dimensions, as for microstrip transmission lines. No RF currents are allowed over the third spatial axis (i.e., in or out of the layer);
- 2.5D means that while RF currents are considered in two axial dimension, fields are evaluated in all three spatial dimensions;
- 3D planar considers currents and fields over all the spatial directions, but circuits are layered over stratified dielectric media, as occurs for PCB circuits;
- 3D arbitrary or full 3D considers electromagnetic interactions occurring over all spatial directions.

In the power electronics system, the following EDA products were evaluated and used to apply the PCB analyses listed above:

- PEPro [30] and Advanced Design System (ADS) [31] from Keysight Technologies;
- Sigrity PowerSI [21] and Allegro PSpice System Designer [22] from Cadence;
- ANSYS SIwave [23] and ANSYS Electronics Desktop/Q3D extractor [24].

All mentioned tools were analyzed in order to simulate PCBs’ design for electronics systems aimed at power conversion. This led to imposing limitations on the frequencies of the signals used.

The software was evaluated by considering the same case study characterized by a PCB with a single layer of dimensions 70.43 mm x 129.52 mm made of Alumina 92 pct, with a thickness of 0.1 mm, and copper tracks of 0.17 mm. Signals in a bandwidth up to 12 kHz were considered.

An example of the workflow adopted to evaluate each software is reported, as an example, for the specific case of ANSYS Q3D extractor in Figures 1–3. The first step is importing the model (Figure 1), then the materials must be defined (Figure 2), especially determining the conductive areas, and, finally, defining the sources and sinks of the structure (Figure 3).

Figure 1. Importing in Q3D Extractor the entire system on which to operate the PCB simulation.
The main features of 2.5D simulation tools are summarized in Table 2.

Table 2. Comparison of the main features software for 2.5D electromagnetic simulation.

| Features                  | Keysight Tech. PathWave ADS | Cadence Allegro PSpice System | PSpice System Designer | ANSYS Electronics Desktop | ANSYS SIwave |
|---------------------------|-----------------------------|-------------------------------|------------------------|---------------------------|--------------|
| High-Frequency Solver    | •                           | •                             |                        |                           |              |
| FEM Solver Method         | •                           | •                             | •                      |                           |              |
| MoM Solver Method         | •                           | •                             | •                      |                           |              |
| FDTD Solver Method        | •                           | •                             | •                      |                           |              |
| Transmission Line RLGC   | •                           | •                             | •                      |                           |              |
| Parameter Extractor      | •                           | •                             | •                      |                           |              |

The main features of 3D simulation tools are summarized in Table 3.
In fact, the field of power conversion implements signals whose frequency does not exceed hundreds of kilohertz; however, the switching signals applied to the power elements may have rising edges so as to generate harmonics in the order of 100 MHz, which can compromise the integrity of the signals and which must be taken into account in order to evaluate the parasitic effects. This means that the choice of the integration step and its variability are crucial aspects, not always efficiently tackled by each software, as in Cadence products, which are optimized for specific bandwidths.

Another aspect that was taken into consideration concerns the capability that the tool has to interface with the flows coming from other applications used by some design teams of the manufacturing company for the PCB design. This parameter is of fundamental importance in order to evaluate any necessary reworking to transfer a project from one tool to another when moving from one design stage to the other.

The simulation times and the necessary computational load, which are obviously related to the required hardware, represent other important aspects that must be taken into consideration, since they lead to an increase in costs regarding licenses and the use of computational grids to achieve the required goal.

These aspects are summarized in Table 4.

Table 3. Comparison of the main features software for 3D electromagnetic simulation.

| Features                        | Keysight Tech. PathWave ADS | Cadence Allegro PSpice System Designer | ANSYS Electronics Desktop | ANSYS SIwave |
|---------------------------------|-----------------------------|----------------------------------------|----------------------------|--------------|
| Import from EDA Tool            | •                           | •                                      | •                         | •            |
| Quasi-Static Solver:            | •                           | •                                      | •                         | •            |
| Low Frequency                   | •                           | •                                      | •                         | •            |
| High-Frequency Solver           | •                           | •                                      | •                         | •            |
| FEM Solver Method               | •                           | •                                      | •                         | •            |
| MoM Solver Method               | •                           | •                                      | •                         | •            |
| FDTD Solver Method              | •                           | •                                      | •                         | •            |
| S-, Y-, Z-Parameters            | •                           | •                                      | •                         | •            |

Performing time-domain cosimulations, all packages offer their own optimized and integrated workflow, which, according to the user experience, appears to be more appropriate as a well-integrated environment, allowing passing the parameters obtained in the frequency domain (S-parameter model) to the electronic circuit to be simulated in a few steps.

For each software, multiple solver interfaces can result in being more intuitive to use than the others, and this could be an added value for the choice of the preferred package.

In our use cases, when interested in low-frequency parasitic extraction, ANSYS Q3D Extractor appeared to be an adequate tool for quasi-static analysis. From another point of view, when interested in studying power and signal integrity problems, Cadence Sigrity and ANSYS SIwave have suitable offered methods. Keysight PEPPro turned out to be an optimized tool for power conversion analysis, which can be largely used for time-domain circuit cosimulation.

Therefore, each tool presents different skills on the basis of the critical issue that has to be analyzed. However, a suitable combination of the tools seems to show some
effectiveness in tackling the problem posed by PCB simulation for power conversion when considering the needs of the developer and the manufacturer.

As concerns the release of novel versions of the software, this may have an impact on software performance, as well as updating the hardware on which the simulations are run, but there are specific features that are rooted in the structural design of the software package and the related theoretical basis on which numerical methods are defined. These aspects will likely remain in all the future developments of the given software.

Nevertheless, looking beyond what the current possibilities of analysis and investigation offer, new perspectives can be opened by exploiting the aforementioned tools. The board model is defined as a set of partial differential equations in time domain and in the 3D spatial domain. Of course, for each spatial point, a dynamical system that describes the EM behavior of the board exists. As regards this model, it could be conceived of in the context of the cellular nonlinear network (CNN) paradigm [32,33], where time-varying and space-dependent templates are considered. This means conceiving of a generalized CNN in which templates are chosen with respect to the different materials and their modification in time as a consequence of the operating regimes. In this way, it would be possible to cope with spatiotemporal time-varying nonlinear phenomena, such as those occurring in a complex PCB with power-integrated devices. Therefore, in this perspective, the CNN approach can be appealing for addressing, with a new paradigm, the mathematical modeling of the PCB board.

5. Case Study Analysis in ANSYS Q3D Extractor

Following the considerations drawn from the previous comparative analysis, we selected a particular case of interest for the manufacturer and applied ANSYS Q3D Extractor since, for the specific purpose, it appeared to be the most reliable solution.

5.1. The Case Study

The STEVAL-CTM009V1 evaluation kit for motor control is designed to highlight the performance of ST Power MOSFETs based on STripFET F7 technology. The STEVAL-CTM009V1 kit is composed of the STEVAL-CTM004V1, STEVALCTM005V1, STEVALCTM006V1, and STEVAL-CTM008V1 boards used for an inverter power stage for three-phase motors, as reported in Figure 4.

Figure 4. STEVAL-CTM009V1 board stack.

The STEVAL-CTM004V1 contains the following parts:

- Insulated metal substrate (IMS);
- Hosts 36 STH310N10F7 or STH315N10F7 power MOSFETs in the $H^2PAK - 6\times$ switch) package;
- Decoupling gate resistors (2.2 $\Omega$);
5.2. Extraction Scenario

In this analysis, we were interested in obtaining the parasitic extraction for some specific frequencies. Specifically, two analysis conditions were assumed for the system:

- DC analysis;
- AC analysis. In the following, in AC, it was assumed that the system works at a frequency of 12 kHz.

In particular, we focused on the extraction of values related to the parasitic resistance and the parasitic inductance of three areas of the system, namely:

- DC+/DC−/phase net;
- High-side gate driver net;
- Low-side gate driver net.

The choice of these three paths was dictated by the indications of the stakeholders, which in our case were the developers and design engineers. This expresses the need to know the resistance and inductance values that affect circuit paths in which anomalous values of these parameters can cause serious system malfunctions as a high resistance, or inductance, value, on the gate driver nets can cause serious problems in terms of the integrity of driving signals and/or the power integrity problem. It must be remembered that for each leg of the system, there is a large number (12) of Si-MOSFET devices and that they can switch at different times, due to the delays that propagate along the path.

On the DC/DC/phase net, given the high load currents that flow through it, the presence of noncongruent parasitic resistance values can determine the onset of points with high temperatures that can lead to system failure.

5.3. Importing Files into Q3D Extractor

The following type of files can be imported into Q3D Extractor: 2D model (GDSII) files, 3D model files, and DXF and DWG format files.

In particular, a file with the .stp extension was imported, allowing importing the geometry of the system without any information about the materials. This led to a significant reworking phase, which saw the assignment of materials and, in some cases, the redefinition of the geometries in order to select the parts of the system on which parasitic extraction must be performed.

The steps characterizing this phase of the study are summarized in Figure 5.
Given the functional symmetry displayed by the considered system, it was decided to examine only one of the three legs that compose the inverter power stage for the three-phase motors.

5.4. Assigning and Editing the Material Properties for the Geometric Elements

For each geometric object involved in the EM simulation process, a material must be assigned. In relation to the functionality related to each geometry, the following materials were used in our system: Alumina 92 pct, brass, and copper.

An overview with the assignment of the materials and the related geometries involved is shown in Figure 6.

![Figure 6. Complete assignment of the materials.](image)

Note that among the elements, there is a class of objects classified as nonmodel. A nonmodel object is an object that does not affect the geometric model and the solution process, as it is used for analysis only.

5.5. Interconnections and Device Creation

As mentioned above, the workflow chosen to directly import the system into Q3D Extractor involved a considerable reworking both for the definition of the geometries and for the assignment of the materials.

At the end of this first phase, in relation to how the geometries were created in the .stp file, it was necessary to carry out the meticulous work of redefining the geometries for the creation of the interconnections and components.

Figure 7 summarizes the steps and instructions present in the Q3D Extractor 3D editor to obtain the interconnections and devices on which to perform the EM simulation in order to obtain the corresponding values of the parasitic parameters.

![Figure 7. Elementary geometric objects must be properly joined to create board interconnects and components.](image)
5.6. Setup Sources, Sinks, and Nets

To setup the analysis for parasitic resistance and inductance, at first, it is necessary to define nets' source and sink terminals. Attached to each source terminal, there is an independent current source.

The sink terminal collects all the currents injected at the source terminals and allows the flow out of the conductor back into the independent sources, closing the electrical circuit.

A net is a collection of contact conducting objects separated by nonconducting or background material. The final step is to define the distinct nets in the problem.

5.7. Specifying Solution Settings and Generating Reports in Q3D Extractor Projects

The Solve Setup window of Q3D Extractor allows specifying how the solution will be computed. More than one setup per design can be made, and each solution setup includes the general data related to the solution generation and the adaptive mesh refinement parameters for the areas where the highest error is retrieved.

When an adaptive analysis is performed, Q3D Extractor generates a field solution using the specified mesh. The system computes the desired parameters and the changes with respect to the previous step (the so-called delta matrix). It then compares the delta with the percent error. When the delta matrix is less than the specified percent error value, the field solution process stops.

The simulator performs an error analysis in each triangle (capacitance and AC analysis) or tetrahedron (DC analysis) in the mesh. Elements with the highest energy error are then refined (broken down into smaller elements), producing a more accurate solution in these areas.

Another solution is generated using the refined mesh, and the system repeats this process until the stopping criteria are satisfied.

5.8. Parasitic R-L Extraction Results

After completing the solution, we chose to view and analyze the results through matrices computed for resistance and inductance during each adaptive or nonadaptive solution. The matrices refer to all the possible paths that can be defined within the net between the sources and the sink. The frequency at which the data were obtained and the unit of measurement are reported. It should be noted that not all the results provided are essential for the purpose of the project. In relation to the nets defined by the tables, we report in Figure 8 the data of interest, i.e., the parasitic resistance and inductance values, both in the AC and DC analysis, \( R_{ij} \) and \( L_{ij} \), between specific points labeled in Figure 9.

![Figure 8](image-url)

**Figure 8. Cont.**
Without going into detail on how Q3D extracts the values of parasitic parameters, making a quick examination between the values returned by the simulation and the circuit topology, we can understand qualitatively if there is a congruence with the one obtained. Only the laboratory can establish the correctness and congruity of the results during the implementation phase, which was in parallel and followed this simulation activity.

Figure 8. Estimation of parasitic resistance and inductance: (a) AC parasitic resistances, (b) AC parasitic inductances, (c) DC parasitic resistances, and (d) DC parasitic inductances.

5.9. Discussion

The considered scenario was based on a configuration with two nets, DC+ and DC−, activated by a source and a sink, located on the most extreme points, while several other source points were present on the net PH. This is the most critical scenario that can be configured as regards the net PH.

In this case, we simulated the condition in which the net PH not only was the medium of the currents flowing to and from the switching power devices, but given its defined position on the symmetry line of the system, it was affected by the presence of both DC+ and DC−, when these nets were crossed by currents.

These boundary conditions were reflected both in the values of the parasitic resistances and inductances extracted in DC and in AC, which were identified in precise sections of the net defined by points of the median line of PH. The points, between which the parasitic values of R and L were determined, precisely mark the topology of the net, positioning themselves between the source and the drain of six pairs of power MOSFETs in the H2PAK-
6 (6× switch) package with the source on DC− (low side) and the drain on DC+ (high side), respectively.

According to this configuration, the net PH turns out to be the seat of the total current flowing in the leg under examination of the three-phase converter.

The current values, in relation to the load, can be very high if it is taken into account that the switching devices used can support, from the specifications, currents of 180 A at 100 V, even if the board is designed for low-voltage uses (up to 48 V). Even if the switching frequency of the system under examination is not very high, the reactive aspects of the impedances that appear in every section of the net were not neglected and exerted their influence where voltages and currents commuted with a certain speed and with rather steep rising edges, generating “anomalies” in the waveforms of the voltages involved, such that the electrical quantities assumed values outside the specifications.

Under certain conditions, the voltage and current variations that were generated were capable of triggering severe malfunctions that could lead to damage to the system. Generally, these overvoltages can lead to early switching on of the device or to delayed switching off of the switching device, generating short circuits between the low and high side of the system. In the power section, malfunctions linked to undervoltages can alter the values of the electrical output quantities, not ensuring the right power required by the load.

From this point of view, the indications provided to the designers allowed improving not only the dissipation of power in every single section of the net, but also the interactions of the various inductances, taking into account the contribution provided by each single component, overlooking the affected stretch of the net.

This resulted in a series of indications that led the layout engineers to optimize the geometry of the PCB tracks, allowing them to identify the most critical points in terms of sustaining the current and consequent thermal dissipation, thus providing a more rational arrangement of the power devices on the PCB tracks and identifying the delay or disturbance factors for switching. The efficient integration between the driving tracks of the power devices and the PCB tracks designed to support the electrical power to be transferred to the load allowed reducing the mutual interaction with the consequent improvement of the signal integrity.

### 6. Conclusions

In conclusion, this work gave an overview of the problems and the software simulation tools related to the PCB analysis for power electronics applications, with particular attention to the perspective of the developers and manufacturer, in order to provide assistance during the design phase of the device.

Different software environment/packages that support the PCB analysis were highlighted, and the comparison between the most popular and reliable tools, providing direct assistance to the costumer, especially when it is a leading company in the semiconductor market, was performed especially for circuit simulation, electromagnetic 2.5D analysis, and 3D analysis.

The results of the comparative study on the mentioned tools was reported in order to better understand which one best suits the specific needs and the type of analysis that is intended to achieve during the design phase, in order to ensure a higher reliability and timeliness to stakeholders and costumers.

The mathematical details related to the analyzed solutions were not discussed in detail due to the different scope that was pursued in this technical note. In fact, including more details and descriptions of the algorithmic solutions implemented by commercial software companies is outside the scope and the aims of the work described in this technical note.

The evaluation of the parasitic dynamics allows integrating in the simulation phase of the design process useful information that prevents unwanted behavior and sensibly improves the testing phase of the real device. Leading companies in the semiconductor market, in fact, have the need to propose to their stakeholders systems that guarantee qual-
ity in terms of electromagnetic compliance, energy savings, and reliability, together with the spatial optimization of the boards. The analysis proposed in this technical note allows such leading companies to shorten the design phase, maintaining a high level of performance, thus meeting the standards imposed by the fast dynamics of the semiconductor market.

Choosing the most adequate software solution for innovative designs and projects, in fact, means, for a big company, planning and performing a strategic investment. In particular, the analysis proposed here allows identifying the most suitable solutions for the evaluation of boards, responding to timely performance tasks, such as small size, power handling, and integration of a high number of packages in the same board. A semiconductor company must consider this choice as a mission in order to give to the board designers the suitable tools allowing them to save time and to enhance the reliability, by exploiting the interaction between the developer of the software and the designer of the device and becoming prompt on the market. Therefore, the classic device design software and the board design software must be jointly considered to realize competitive and innovative electronic systems in industrial applications. This study was hence mainly addressed at giving integrated information to perform the correct choice in accordance with the company policies. Of course, this point of view complements the classical approach of using software for testing and validating boards and devices when they are already physically realized.

Author Contributions: Conceptualization, M.B., A.B., L.F., C.F., M.F., A.C., G.R. and G.V.; methodology, C.F., G.R. and G.V.; validation, C.F., G.R. and G.V.; formal analysis, L.F., M.B., A.B. and M.F.; investigation, C.F., G.R. and G.V.; resources, A.C., G.R. and G.V.; data curation, C.F. and G.V.; writing—original draft preparation, M.B., A.B., L.F., C.F., M.F., A.C., G.R. and G.V.; writing—review and editing, M.B., A.B., L.F., C.F., M.F., A.C., G.R. and G.V.; supervision, A.C., L.F., M.B., A.B., M.F. and G.R.; funding acquisition, L.F. All authors have read and agreed to the published version of the manuscript.

Funding: This research has been financially partially supported by STMicroelectronics under the contract “Utilizzo di metodologie di simulazione di sistemi hardware e software basate su Virtual Platform a supporto della ricerca, la progettazione, la verifica e lo sviluppo di sistemi complessi basati su dispositivi elettronici ST” and by the MiSE project M9 (CDS 000448)—CUP C32F18000100008.

Institutional Review Board Statement: Not applicable.

Informed Consent Statement: Not applicable.

Conflicts of Interest: The authors declare no conflict of interest.

References
1. Gole, A.M.; Nayak, O.B.; Sidhu, T.S.; Sachdev, M.S. A graphical electromagnetic simulation laboratory for power systems engineering programs. *IEEE Trans. Power Syst.* 1996, 11, 599–606. [CrossRef]
2. Chen, Z. PCB Microstrip Line Far-End Crosstalk Mitigation by Surface Mount Capacitors. In Proceedings of the 2019 IEEE 69th Electronic Components and Technology Conference (ECTC), Vegas, NV, USA, 28–31 May 2019; pp. 1989–1995.
3. Li, J.; Shrivastava, P.; Gao, Z.; Zhang, H.C. Printed circuit board recycling: A state-of-the-art survey. *IEEE Trans. Electron. Packag. Manuf.* 2004, 27, 33–42.
4. Buscarino, A.; Fortuna, L.; Famoso, C. Reduced order thermal models for electronic devices. In Proceedings of the 2020 IEEE 20th Mediterranean Electrotechnical Conference (MELECON), Palermo, Italy, 6–18 June 2020; pp. 254–258.
5. Mirotznik, M.S.; Prather, D. How to choose electromagnetic software. *IEEE Spectr.* 1997, 34, 53–58. [CrossRef]
6. Danković, D.; Vračar, L.; Prijić, A.; Prijić, Z. An electromechanical approach to a printed circuit board design course. *IEEE Trans. Educ.* 2013, 56, 470–477. [CrossRef]
7. Pace, L.; Idir, N.; Duquesne, T.; De Jaeger, J.C. Parasitic loop inductances reduction in the PCB layout in GaN-based power converters using S-parameters and EM simulations. *Energies* 2021, 14, 1495. [CrossRef]
8. Alvarez, C.; Murali, P.; Swaminathan, M.; Oishi, Y.; Takashiro, J.; Nagatsuka, R.; Watanabe, N. Demonstration of a High-Inductance, High-Density, and Low DC Resistance Compact Embedded Toroidal Inductor for IVR. In Proceedings of the 2021 IEEE 71st Electronic Components and Technology Conference (ECTC), San Diego, CA, USA, 1–4 June 2021; pp. 1293–1299.
9. Ho, W.; Ji, P. *Optimal Production Planning for PCB Assembly*; Springer Science & Business Media: Berlin/Heidelberg, Germany, 2006.
10. Zhang, Y.; Li, J.; Wang, J. Investigations on driver and layout for paralleled GaN HEMTs in low voltage application. *IEEE Access* 2019, 7, 179134–179142. [CrossRef]
11. Hofer, M. New approach for faster thermal modeling of PCBs based on power module applications. In Proceedings of the 2019 IEEE International Workshop on Integrated Power Packaging (IWIPP), Toulouse, France, 24–26 April 2019.

12. Jørgensen, A.B.; Cheng, T.H.; Hopkins, D.; Beczkowski, S.; Uhrenfeldt, C.; Munk-Nielsen, S. Thermal characteristics and simulation of an integrated GaN eHEMT power module. In Proceedings of the 2019 21st European Conference on Power Electronics and Applications (EPE’19 ECCE Europe), Genova, Italy, 2–5 September 2019.

13. Lin, H.N.; Tseng, W.D.; Wu, C.H.; Yeh, T.H.; Ho, T.H. Root cause analysis and defect ground effect of EMI problem for power electronics. In Proceedings of the 2019 Joint International Symposium on Electromagnetic Compatibility, Sapporo and Asia-Pacific International Symposium on Electromagnetic Compatibility (EMC Sapporo/APEMC), Sapporo, Japan, 3–7 June 2019; pp. 440–443.

14. Montrose, M.I. Printed Circuit Board Design Techniques for EMC Compliance; IEEE Press: Piscataway, NJ, USA, 1996; Volume 1, p. 996.

15. Harrington, R.F. Field Computation by Moment Methods; Wiley-IEEE Press: Hoboken, NJ, USA, 1993.

16. Gibson, W.C. The Method of Moments in Electromagnetics; Chapman and Hall/CRC: Boca Raton, FL, USA, 2007.

17. Okereke, M.; Keates, S. Finite Element Applications: A Practical Guide to the FEM Process; Springer: Berlin/Heidelberg, Germany, 2018.

18. Gedney, S.D. Introduction to the finite-difference time-domain (FDTD) method for electromagnetics. Synth. Lect. Comput. Electromagn. 2011, 6, 1–250. [CrossRef]

19. Ikeda, H.; Itoh, Y. PCB-Based Cross-Coupled Differential VCOs Using a Novel LC-Tank Comprised of the Chip Inductors. IEICE Trans. Electron. 2018, 101, 744–750. [CrossRef]

20. Mitzner, K. Complete PCB Design Using OrCad Capture and Layout; Elsevier: Amsterdam, The Netherlands, 2011.

21. Sigtry PowerSL. Available online: https://www.cadence.com/en_US/home/tools/ic-package-design-and-analysis/si-pi-analysis-point-tools/sigtry-powerisl.html (accessed on 17 November 2021).

22. Cadence Allegro Pspice. Available online: https://www.cadence.com/ko_KR/home/tools/pcb-design-and-analysis/analog-mixed-signal-simulation/allegro-pspice-simulator.html (accessed on 17 November 2021).

23. ANSYS Electronics Desktop. Available online: https://www.ansys.com/products/electronics/ansys-electronics-desktop (accessed on 17 November 2021).

24. ANSYS Siwave. Available online: https://www.ansys.com/it-it/products/electronics/ansys-siwave (accessed on 17 November 2021).

25. Arrillaga, J.; Arrillaga, J.; Smith, B. AC-DC Power System Analysis (No. 27); IET: London, UK, 1998.

26. Bogatin, E. Signal and Power Integrity-Simplified: Signa Integ Simpl Secon E2; Pearson Education: London, UK, 2009.

27. Brooks, D.; Adam, J. PCB Design Guide to Via and Trace Currents and Temperatures; Arttech House: London, UK, 2021.

28. Archambeault, B.R.; Ramahi, O.M.; Brench, C. EMI/EMC Computational Modeling Handbook; Springer Science & Business Media: Berlin/Heidelberg, Germany, 2012; Volume 630.

29. Sanjay, D.; Varadarajan, S. EMI-EMC Analysis; Lambert Academic Publishing: Chisinau, Moldova, 2019.

30. KEPro. Available online: https://www.keysight.com/it/en/product/W2376EPE/kepro-momentum-fem-element.html (accessed on 17 November 2021).

31. Keysight Technologies PathWave ADS. Available online: https://www.keysight.com/it/en/products/software/pathwave-design-software/pathwave-advanced-design-system/pathwave-ads-simulation-elements.html (accessed on 17 November 2021).

32. Buscarino, A.; Fortuna, L.; Frasca, M. Essentials of Nonlinear Circuit Dynamics with MATLAB and Laboratory Experiments; CRC Press: Boca Raton, FL, USA, 2017.

33. Arena, P.; Bucolo, M.; Buscarino, A.; Fortuna, L.; Frasca, M. Reviewing Bioinspired Technologies for Future Trends: A Complex Systems Point of View. Front. Phys. 2021, 9, 597. [CrossRef]