Design Tools for Power Electronics: Trends and Innovations

Uwe DROFENIK[*], Didier COTTET[**], Andreas MÜSING[*] and Johann W. KOLAR[*]

[*] Power Electronic Systems Laboratory, ETH Zurich, ETH-Zentrum / ETL H13, CH-8092 Zurich, Switzerland
Phone: +41-1-632-4267, Fax: +41-1-632-1212, E-mail: drofenik@lem.ee.ethz.ch
[**] ABB Switzerland Ltd, Corporate Research, CH-5405 Baden-Dättwil, Switzerland

Abstract: Numerical simulation is a standard procedure in the design of power electronic systems. With simulation, one can test new concepts immediately without the need to order components and assembling which might be time-consuming and expensive. If something fails, there is no destruction but information about too high voltages and/or currents. Critical operating states just before failure can be exactly reproduced, and currents, voltages and junction temperatures can be easily monitored in simulation which makes it comparably easy to identify problematic designs. Expensive equipment for measurement, power supply and load which is essential for testing prototypes is not needed in a first design stage. Further advantages of simulation are the ability to easily visualize fields, flows and distributions of physical properties, and the ability of automated parameter optimization and/or statistical analysis with Monte Carlo techniques.

Due to these advantages it would be desirable to replace designing and testing prototypes by numerical simulations as far as possible in order to reduce development time, save development cost and detect reliability problems. Unfortunately, practical simulation will never fully map reality. The power electronic system under investigation has to be simplified in order to be able to handle the model with a computer. Numerical simulation will always give a result, but it is up to experience and knowledge of the design engineer to verify the usefulness and/or accuracy of the result.

In the paper we discuss what can be numerically simulated, what limits are given to modelling by scaling laws and what kind of developments we might experience in the future. Emphasis is on the numerical simulation of converter systems.

Keywords: Numerical simulation, multi-disciplinary simulation, PEEC, thermal, EMI, reliability

2. Introduction

1.1 Trends and Requirements

What do we want to simulate in power electronics?

- ***Circuits**:* The design engineer wants to calculate current- and voltage-waveforms of a converter under different operating conditions verifying functionality. All component voltage- and current-levels have to remain within safe value ranges.
- ***Thermal Design / Cooling System:*** The design engineer is interested in transient semiconductor junction temperatures under different operating conditions (e.g. temporary overload condition), transformer and inductor temperatures, and the performance of different possible cooling systems employing water- or air cooling and/or heat pipes.
- ***EMI-Filter Design:*** It would be desirable to be able to calculate CM- and DM-noise of the converter system based on the PCB design and simplified models of the semiconductor switches before building and testing an EMI-filter.
- ***Inductive Components:*** Inductors and transformers are key components in power electronic systems. They significantly contribute to the system losses, system volume and weight. With increasing emphasis on converter optimization, there is a growing desire to employ more realistic non-idealized inductor models in circuit simulations considering iron losses, eddy-currents, skin- and proximity-effects, and thermal behavior.
- ***Optimization:*** Depending on application, power electronic systems have to be optimized in terms of costs, volume, and/or weight. As example, a design task might be to minimize the heat-sink volume, to minimize the EMI filter weight, to maximize the reliability of the power module, or to maximize the converter's robustness for a given range of operating conditions.
- ***Reliability / Life Time:*** According to a study in the electronics industry [1], a significant percentage of all failures in electronic systems are related to temperature issues. Therefore, calculating module and/or converter system reliability based on transient temperatures will become an important feature of power electronics simulation software.

What do we demand from simulation software?

- ***Fast processing:*** Typically, the simulation time should be limited to a couple of hours, because often a large number of simulations is needed to investigate and verify a design before building a prototype. The faster the simulation, the more effective the simulation-based design phase.
- ***High accuracy:*** Simulators will always give results in terms of time-dependent values, but in case of poor modelling the simulated results will have nothing in common with reality which makes them useless. The design engineer needs experience and good knowledge in modelling and simulation in order to guarantee a certain accuracy of the results. If the simulation software is augmenting the task of setting up realistic models, or provides warnings in case the model and/or the simulation results look faulty, this will significantly improve the quality of this design phase, and will help preventing non-working experimental prototypes.
- ***Multi-Disciplinary:*** Power electronic applications cover a wide range of frequencies (DC to GHz), dimensions (um to km), temperatures (-55°C to 275°C) and power (mW to MW), and therefore require design tools with extremely broad numerical capabilities. Power electronics is located at the intersection of

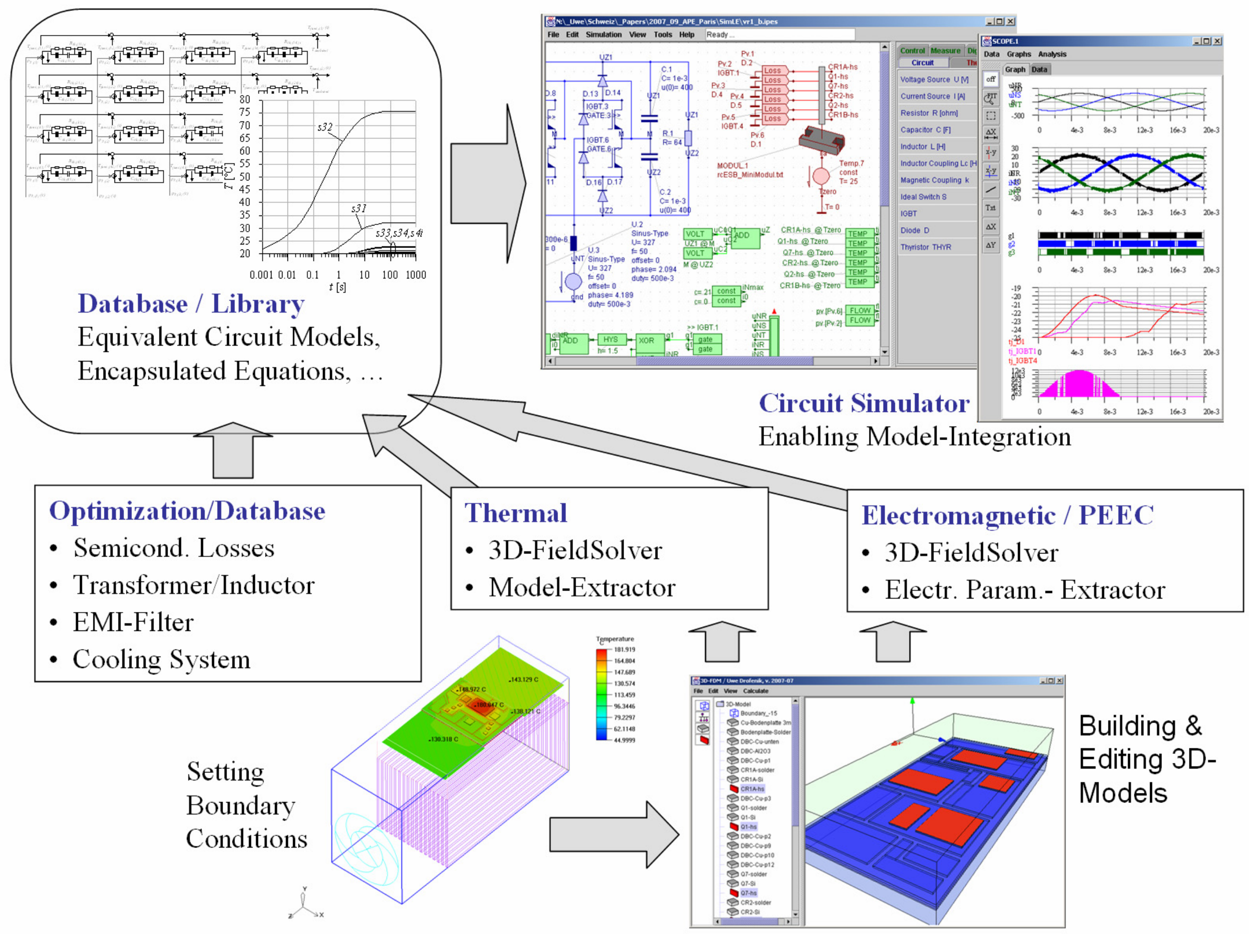


Fig.1. Basic structure of the Power Electronic Virtual Design Platform under development at the Power Electronic Systems Laboratory (PES), ETH Zurich. In a 3D-editor three-dimensional models of structures (e.g. a power module as shown in the figure) can be defined. A built-in thermal FEM solver extracts thermal equivalent circuit models that can be stored in a database/library for flexible use by the Circuit Simulator. In a similar way, another built-in solver (PEEC - electromagnetic) extracts electric parameter networks. In the figure, the thermal circuit within the Circuit Simulator is shown in red color, and the symbol of the power module hides the extracted thermal equivalent network generated by the 3D Thermal Solver. Other tools like heat sink optimization, EMI filter design, transformer/inductor modelling and design, cooling system design and reliability calculations are provided by a tightly integrated Optimization Toolbox/Database.

various different engineering disciplines, especially circuit design, thermal design (heat transfer, fluid dynamics), electromagnetics (EMI, inductors, transformers), digital electronics and DSP-programming, control theory, material science especially with focus on packaging, and reliability engineering. This multi-disciplinary aspect of power electronics becomes increasingly significant with always rising switching frequencies, need for higher power densities, and demand for higher reliability and robustness in wide parameter ranges. Accordingly, future simulation software has to reflect this multi-disciplinary aspect, and should provide the ability of performing coupled simulations. Because temperature has major impact on reliability, we regard especially thermo-electric, thermo-magnetic, and thermo-mechanical to be of importance.

- ***Easy-to-use:*** Because of the increasingly multi-disciplinary character of power electronics, a design engineer educated in electrical engineering will face, e.g., the challenge to design a cooling system for a converter, an EMI-filter, and estimate the life time of the power semiconductor modules. Simulation software for performing specialized calculations, e.g., three-dimensional temperature fields in power modules, need to have a user interface that is simple enough to be handled by an engineer who is not a specialist in this certain field, but should still be powerful enough to give useful simulation results. Furthermore, such software needs interfaces so that data can be exchanged easily between different programs. For example, the thermal simulator giving a thermal network model of a power module must provide the ability to directly export this network into a circuit simulator, so that transient junction temperatures can be calculated.
- ***Augmenting design tasks:*** Optimizing a heat sink for minimum volume at a given thermal resistance is a challenging task but essential if the converter system has to be designed for maximum power density. Although important, such a heat sink optimization is not part of the typical education in electrical engineering. This provides an example of a design task that can be aided by software with an easy-to-use graphical user interface. Another example is the design of an EMI filter which can be augmented by software.

1.2 Multi-Disciplinary Simulation in Power Electronics

Simulation software useful for power electronics exists for single disciplines like circuits and controls (e.g. [2]-[8]), heat transfer and fluid dynamics (e.g. [9], [10]), high-frequency electromagnetics (e.g. [11], [12]) or electromagnetic fields (e.g. [13], [14]). What is missing is an efficient coupling in order to take into account multi-disciplinary aspects of power electronics. Software to realize general coupled simulations is commercially available (e.g. [15]), but difficult to handle for non-scientists and/or limited in performance. It might be that power electronics is a comparably small discipline within electrical engineering so that software companies don't see their main business in developing specific sophisticated software. On the other hand, there seems to be a very large group of smaller power electronics companies that employ simulation software to very limited extent because of the costs of the software licence and the difficulty to employ specialists being able to handle different simulation tools efficiently. To keep up with the general trends of increasing power density, increasing switching frequencies and the demand for improved reliability, there will be strong requirement for software being able to handle multi-disciplinary simulations.

Transient power electronic circuit simulations for PWM rectifiers typically run over a few mains periods lasting several tens of milliseconds. Today switching frequencies are typically in the range of 50kHz to 200kHz with the tendency to rise due to improved semiconductors. Numerical time steps have to be typically by a factor 100 smaller than the smallest time constant in the system resulting in a numerical time step width smaller than 100ns for 100kHz switching frequency. Therefore, over one mains period (20ms) there is a minimum number of 200.000 numerical time-steps required. In case one would like to take into account semiconductor switching behaviour which takes typically several tens of nanoseconds, the numerical step width would have to be reduced even further. If, e.g., thermal coupling has to be considered, a thermal model of a power module and the cooling system has to be set up. If one employs the finite element method (FEM) to calculate the temperature distribution within the 3D-model, one single FEM-simulation takes typically a few minutes up to a few hours. Theoretically, this has to be done at every time-step of the numerical circuit simulation to achieve true coupling. Assuming 20min calculation time for one FEM-simulation, the total time of the coupled simulation would be 200.000 × 20min = 7.6years. Considering that many mains periods are needed to bring the system into thermal steady-state and/or the fact that in some applications there is the desire to take semiconductor switching behaviour into account, this approach of thermal coupling is not practical, even if computational power of computers keeps doubling every few years. Similar problems arise in case of coupling a circuit simulation directly to a 3D-FEM solver calculating electromagnetic effects.

One possible strategy of performing numerically efficient coupled simulations is to extract equivalent circuit models from 3D-FEM simulations (especially thermal and fluid dynamics, and electromagnetics), which are typically represented by small networks that can be integrated easily into the circuit simulator. The insertion of such small equivalent circuit network models will increase the computational effort of the circuit simulation, but the total effort will still be comparably small and could be handled easily. An approach based on such equivalent circuit models is shown in Fig.1 which describes an effort of the Power Electronic Systems Laboratory (PES), ETH Zurich, where a Virtual Design Platform for Power Electronics is currently under development. In some of the following examples we will refer to this simulation platform which allows simulations that are fast, of high accuracy, multi-disciplinary, and easy-to-use. Additionally, it provides tools for augmenting and optimizing certain design tasks typical for power electronic systems research and development.

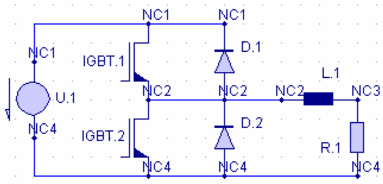


Fig.2. Power circuit topology with three nodes plus ground node.

2. Single-Discipline Simulation

2.1 Power Electronic Circuit Simulation

Each component of the power circuit is described by an equation. Linearizing the equations within one time step of the circuit simulation, results in a system of linear equations that can be solved in matrix form as given by equation (1) as mathematical formulation of the circuit of **Fig.2**.

$$\left(\varphi_{NC1} \quad \varphi_{NC2} \quad \varphi_{NC3}\right)^T \bullet \underline{A} = \underline{b} \qquad (1)$$

A systematic procedure of setting up the simulation matrix is provided by the Node Voltage Method (e.g., p.213 in [16]), where, generally speaking, each node in the power circuit makes one contribution to the order of the simulation matrix **A**. During the simulation, the time proceeds in small steps Δt from t_{START} to t_{END}, and equation (1) is repeatedly solved after each time step. In the example shown in Fig.2, the four nodes of the power circuit result in one simulation matrix **A** of order three because the ground node (*NC4*) potential is set to zero. Accordingly, in Fig.2 we define the number of (matrix-relevant) nodes as $n_{NC} = 3$.

$$CE \propto \quad n_{NC}^3 \qquad (2)$$

As long as the order of **A** is comparably small ($< 10^3$), so-called *direct* matrix solvers can be used. By employing direct solver algorithms like LU Decomposition [17], the computational effort *CE*, characterized by the numbers of executions needed to solve the matrix equation, rises with the third power of n_{NC}, while the order of the matrix is given by n_{NC}. The computation time is approximately proportional to the third power of the number of nodes in the power circuit, and the needed memory space is proportional to the second power of number of nodes [18].

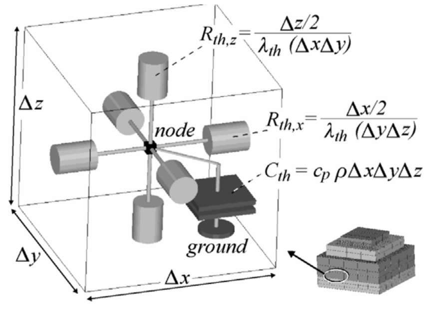


Fig.3. Example of one element for employing FDM.

2.2 Thermal Simulation

With thermal capacitance c_P [Ws/(K·kg)], material density ρ [kg/m^3], temperature dependent thermal conductivity λ [W/Km], thermal flow density w [W/m^3], and temperature T [K], the thermal conduction inside a solid structure is described by the heat conduction equation

$$c_P(T) \cdot \rho \frac{\partial T}{\partial t} = \nabla\left[\lambda(T) \cdot \nabla T\right] + w(\vec{x}, t) \qquad (3)$$

Considering thermal convection, which is the main cooling mechanism in case of water cooling or air-cooled heat sinks there are two additional coupled differential equations (one scalar, one in vector format) named Navier-Stokes [19]. Therefore, considering convection directly will increase the computational effort of this so-called computational fluid dynamics (CFD) problem significantly. A significant improvement provides the strategy to define convection in form of a boundary condition via a so-called heat transfer coefficient and calculate the remaining solid structure employing only heat conduction with no need of CDF simulations. The merits of this strategy are described in detail in [20].

Employing the Finite Difference Method (FDM) or Finite Element Method (FEM), the solid structure is first divided into small elements. Afterwards, equation (3) is linearized within each element's volume. The temperature at the centre point node represents the volume's temperature and the ground node represents ambient temperature as shown in **Fig.3**. Thermal losses within this geometric element would be represented by heat flow injected into the centre node. Thermal resistances and thermal capacitance is defined based on material properties and geometry. Coupled linearized equations of neighbour-elements result in a system of linear equations that can be described in matrix form. The order of the matrix equation is equal to the number of elements. If, e.g., all three sides of a structure are divided by 100, this would result in 1.000.000 elements with a matrix equation of the same order. This shows that in case of 3D-FDM/FEM even small models quickly grow into huge and difficult-to-handle computational problems, which clearly prevents any attempts to solve 3D-FEM problems within each time step of a power electronics circuit simulation.

2.3 Electromagnetic Simulation

The numerical techniques for the solution of field problems can be classified into two groups: Either electromagnetic solvers that are based on the differential formulation of the Maxwell equations (Finite Difference [21] - and Finite Element Methods [22] (FDM, FEM)) or integral equation (IE) based techniques (PEEC [23], Method of Moments (MoM) [24]). A good choice of the appropriate method seriously affects the modeling and simulation effort and the accuracy of the results. The solution variables of FDM and FEM are the field variables *E* and *H*, whereas IE based techniques are formulated in circuit variables (currents and voltages). Bearing in mind a tight integration of the electromagnetics simulation within a circuit solver, IE-based techniques fit better to the concept of multidisciplinary simulations. Furthermore, for IE-based techniques the model handling is more convenient, because only the involved conductor geometries needs a discretization in contrast to FDM or FEM where the whole space including vacuum has to be taken into account. MoM has the huge drawback that it is a high frequency approximation - low frequency to DC cannot be handled correctly by MoM.

The approach used in the Design Platform shown in Fig.1 is the Partial Element Equivalent Circuit (PEEC) method. Software for inductance and capacitance calculation (e.g. FastCap [25], [26], FastHenry [25], [27], FastImp [28], InCa [12]) using the PEEC method has been available for many years, but only recent developments extend PEEC to integrated full wave solver capability which allows to handle electric and magnetic fields, time retardation, dielectrics [29] and non-orthogonal geometries [30].

This makes PEEC a flexible multi-purpose simulation tool which enables to use the same model for frequency domain as well as time domain simulations. PEEC is a full wave Maxwell equation solver, which maps the electromagnetic behavior of arbitrary 3D layouts and geometries into an equivalent network of lumped circuit components. Theoretically, this method gives accurate results from DC up to highest frequencies. The main disadvantages of the PEEC method is that it requires a big amount of computational resources (CPU and memory), but together with an intelligent model generation and subsequent model extraction routine, PEEC is ideally suited for the integration into a multidisciplinary simulation.

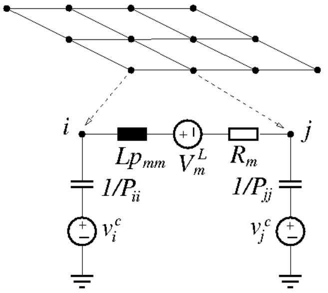


Fig.4. Geometry discretization and PEEC model [31].

In the PEEC method, all regions of interest/conductors are discretized into many partial elements as shown in **Fig.4**. The PEEC method creates matrices of partial inductances L_{pij}, partial coefficients of potential P_{ij} and node resistances R_m. The individual matrices are responsible for the magnetic- and electric field couplings and material conductivity, respectively. The generated lumped circuit network can then be excited by current- and voltage sources, and the frequency- or time domain response is then calculated by a spice-like circuit simulator. The simulation corresponds to a solution of a matrix equation which is generated from the partial element matrices.

2.4 Inductive Component Modelling, Optimization and Augmenting Design Tasks

In order to perform more realistic numerical simulations of power electronic systems, detailed models of inductive components would be very helpful. In [32] and [33] it is shown for a 5kW DC/DC converter how to achieve power densities larger than 10kW/dm^3 by employing an optimization loop with an internal simulation that takes into account detailed inductive component models including iron losses, skin- and proximity-effect, and detailed thermal models. Making such models available to circuit simulation via component databases (e.g. in form of parameterized equivalent network models in combination with equation-based models) would increase the efficiency of the simulation platform significantly. Software tools to create models from 3D-structures and material descriptions will be needed to populate such databases.

In [34] it is shown how to reduce the volume of a heat sink for a 10kW rectifier by a factor five compared to commercially available heat sinks by optimizing the air flow through the fins. For the DC/DC converter described in [33] it was essential to minimize the heat sink in order to achieve power densities up to 10kW/dm^3. Heat sink calculations are typically not covered in electrical engineering education which means that software tools for such optimizations would be very helpful in a power electronics design environment.

Another example of employing simulation in designing cooling systems is shown in **Fig.5**. Here, a converter had to be cooled by forced convection with low cost as the main optimization criterion. Simply by placing heat sink, fan and air-outlet correctly, sufficient cooling could be provided by employing a single fan. Since the air flow inside a housing is typically very complex, it was essential to employ computational fluid dynamics software (CFD). In comparison, designing prototypes based on try and error would be extremely time-consuming. Compared to measurements that were performed at the optimized prototype, the accuracy of the simulation proved to be very good.

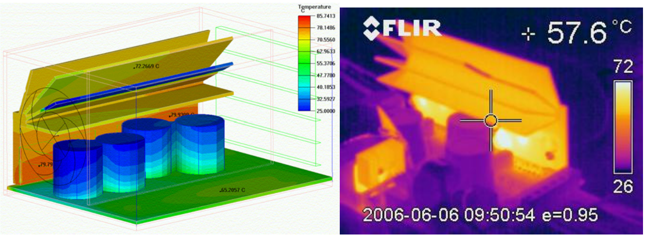


Fig.5. Complex air flow inside a housing is calculated via 3D-CFD-FEM (ICEPAK) and compared to infrared measurement. With calculated 58.6°C at the heat sink centre fin compared to a measurement of 57.6°C, the agreement is within 3%.

Algorithms for complex tasks like EMI-filter optimization are discussed in [35]. Realizing software implementing the algorithms and providing an easy-to-use GUI would be a great benefit because EMI-filter design and optimization is essential in many applications, but often not a core competence of power electronic engineers.

Generally, integrating such tools into a power electronic simulation environment will increase the designer's productivity and the usefulness of the simulations significantly.

2.5 Reliability

In order to guarantee a certain life time for a product it is a strong desire from industry to be able to estimate reliability of converter system and/or subsystems like power modules. As discussed in [1], p.5.4, a significant amount of failures in the electronic industry is due to overheating and temperature-cycling. There are two basic laws describing the impact of temperature on component stress. The Arrhenius model

$$\lambda = \lambda_0 \, exp\left(\frac{-Ea}{k}\left(\frac{1}{T}-\frac{1}{T_0}\right)\right) \tag{4}$$

defines the thermal activation of the failure mechanism [1]. With rising absolute temperature T, the failure rate λ will increase. The Coffin-Manson model describes the stresses on bonding and interface layers as they are typical in power modules as

$$N_f \propto \left(L \cdot \Delta\alpha \cdot \Delta T\right)^{-n} \tag{5}$$

with number of failures N_f, characteristic geometric length L, difference in expansion coefficients $\Delta\alpha$ and temperature cycling amplitude ΔT [1], [36]. With knowledge of the transient temperature during operation it is theoretically possible to calculate a lifetime prognosis as shown in **Fig.6**. Providing lifetime-models within a power electronics simulation platform will be an important add-on. Again here, reliability is usually not part of an electrical engineer's education but will become increasingly important in the future. Accurate and easy-to-use software will be the key to integrate this essential knowledge into the design process.

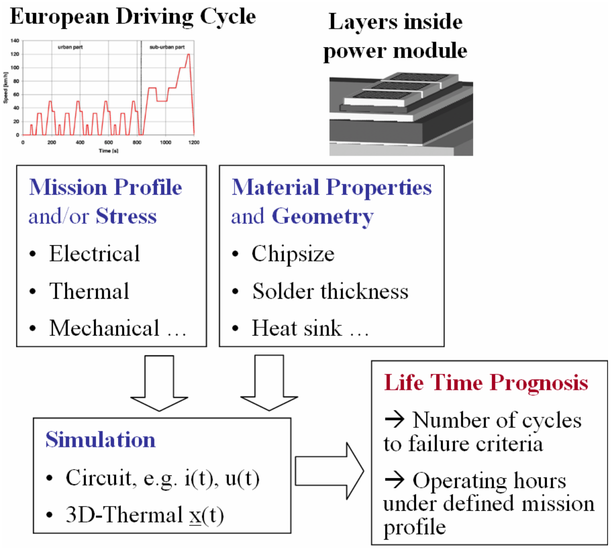


Fig.6. Schematic calculation of estimated reliability for a given stress profile and system properties (geometry, materials, etc.) [37].

3. Multi-Disciplinary Simulation Circuit - Thermal

3.1 Modelling of Thermal Coupling

Power semiconductors placed on heat sinks will heat up neighbour semiconductors by rising the heat sink temperature. Inside power modules the same effect takes place between dies and is generally known as 'thermal coupling'. This effect can be significant, and, therefore, has to be considered in thermal equivalent network models. A general modelling procedure called Impedance Matrix Method [38] is very effective, and will be discussed in the following.

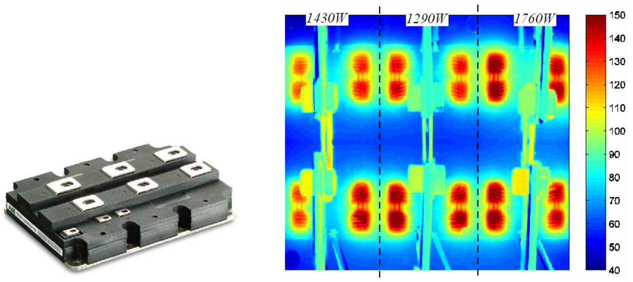


Fig.7. (a) 3300V/1200A ABB HiPak IGBT Module [39] employing 36 internal chips. Here, 24 chips connected in parallel form one switch, and the remaining 12 chips connected in parallel form the anti-parallel diode. **(b)** Stationary temperature distribution for heating all 24 IGBTs. Groups of eight IGBTs connected in parallel equally share a thermal power of 1430W, 1290W and 1760W, respectively. The total thermal power dissipated by the module in this experiment is 4480W (see [20]).

As example, a power module [39] with 36 internal chips is shown in **Fig.7**(a). Performing transient 3D-FEM simulations where a single die is heated via a step-shaped thermal power, all die-temperatures will rise as shown in **Fig.8**. In order to build a thermal model of the power module, each thermal step response gives a simple 3- or 4-stage network-model consisting of thermal resistors and capacitors parameterized via curve-fitting. The models don't have physical meaning, but simply fit the step responses of the power module's internal dies.

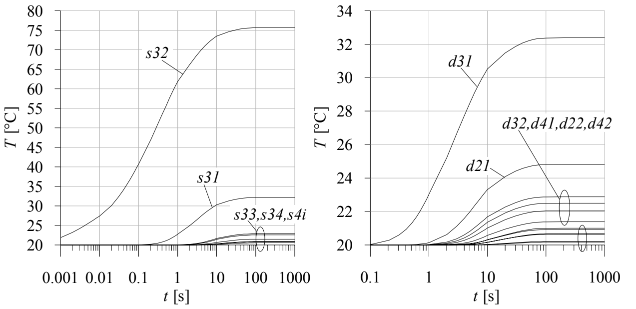


Fig.8. Transient thermal step responses from a 3D-FEM simulation of the power module shown in Fig.7(a), where one single IGBT '*s32*' is heated by a thermal power of 168W resulting in step response temperatures at the centres of all 36 dies labelled s_{ij} and d_{ik}. [20].

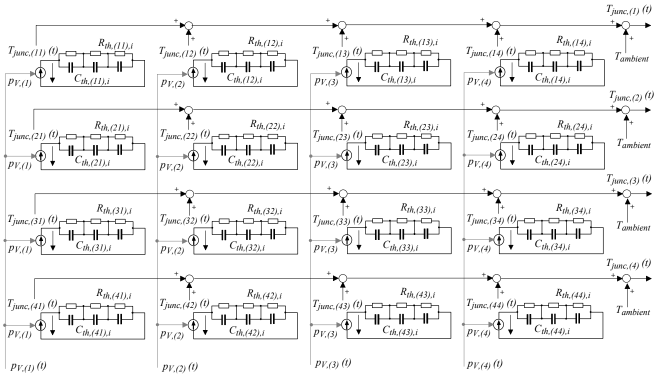


Fig.9. Full dynamic thermal model for only four different chips described by $R_{th}C_{th}$-impedance networks that are coupled via signals $p_{V,(i)}$ and $T_{junc,(i)}$.

In case of 36 thermally coupled dies, there will be a total of 36 × 36 = 1296 step responses to be modelled. Applying superposition to equation (3) is valid under the assumption that material properties are in good approximation temperature-independent. Then, the 1296 network models can be mutually coupled. Such a network model for only four coupled chips is shown in **Fig.9**. The modelling process with emphasis on the accuracy of the 3D-FEM simulations of the thermal step responses is discussed in detail in [20]. Software implementation, scaling issues, coupling with a circuit simulator and numerical simulation are discussed in [18]. Not only for a problem as large as the 36-chip power module, but even for much smaller problems, a careful software implementation of the whole modelling process of the Impedance Matrix Method is by far the best solution in order to be able to handle the problem.

3.2 Switching- and Conduction Losses

Simulation of conduction losses can be simply performed by defining a resistor and a voltage source in series with an ideal switch, where the resistor might be defined as temperature-dependent in order to model the general temperature-dependency of semiconductor losses. For switching losses there are basically two strategies available. One strategy is to try to model the switching process in the time domain with extremely high accuracy and calculate the resulting switching losses. The problem with this approach is that the numerical time steps during the switching action have to be set to very small values which might cause problems with the numerical stability and increase the simulation time significantly.

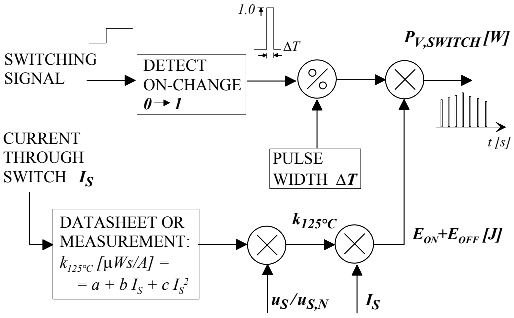
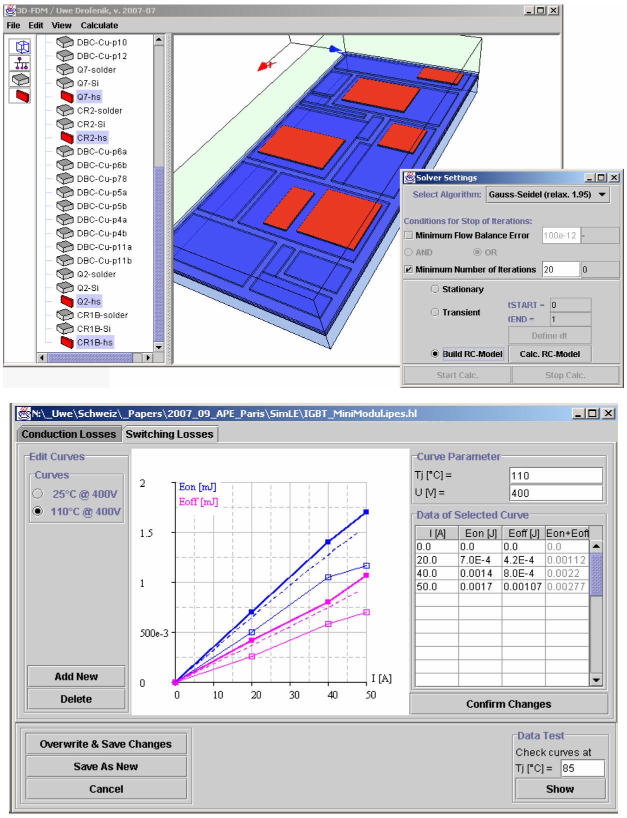


Fig.10: The general scheme employs the current through the switch, the switching signal and the approximation of ratio k_{IGBT} (eq. (9)). It gives the time behavior of the switching power loss $P_{V,SWITCH}$ *(t)* during a numerical circuit simulation. The scheme acts as a simple counter detecting switching-actions.

A different strategy preferable especially for switched system simulations (typical for power electronics) is to implement a counter that injects an energy pulse containing the switching loss energy as given in a datasheet or derived from measurements any time an ideal switch is turned on or off. This will not slow down the simulation speed because the ideal switching action occurs within one numerical time step, the simulation will remain numerically stable, and the accuracy will be very high. Such a simple loss counter scheme as shown in **Fig.10** is described in detail in [40] (see also [41], [42], [43]).

3.3 Transient Junction Temperatures Considering Temperature-Dependency of Semiconductor Losses

An example of an implementation of a coupled numerical simulation circuit-thermal is shown in **Fig.12**. The critical aspect of the coupling is to make the switching- and conduction-losses dependent not only on voltage and current, but also on the transient junction temperatures of the semiconductors. The temperature-dependent loss characteristics given by datasheet and/or measurement of the power module's semiconductors (shown as red plates in **Fig.11**(a)) can be defined via dialog (Fig.11(b)) and attached to the IGBT- and diode-symbols of the power circuit (blue color, Fig.12). The power module is set up as 3D-structure (Fig.11(a)) including the heat sink, and the Thermal Solver will automatically build the full thermal model via impedance matrix (Fig.9) that can be inserted into the Circuit Solver via a file to be attached to the power module symbol (*MODULE.1*, red color, Fig.12). The transient semiconductor losses (temperature-dependent conduction- and switching-losses) are calculated as shown in Fig.10 and can be accessed conveniently from the *Loss*-blocks (six such blocks shown in red color in Fig.12).



Select Algorithm: Gauss-Seidel (relax. 1.95)

Conditions for Stop of Iterations:

☐ Minimum Flow Balance Error 100e-12 -

○ AND ● OR

☑ Minimum Number of Iterations 20 0

○ Stationary

○ Transient tSTART = 0 tEND = 1

● Build RC-Model

○ 25°C @ 400V

● 110°C @ 400V

Curve Parameter

Tj [°C] = 110

U [V] = 400

Data of Selected Curve

I [A]	Eon [J]	Eoff [J]	Eon+Eoff
0.0	0.0	0.0	0.0
20.0	7.0E-4	4.2E-4	0.00112
40.0	0.0014	8.0E-4	0.0022
50.0	0.0017	0.00107	0.00277

Data Test

Check curves at Tj [°C] = 85

Fig.11. **(a)** Dialog-window of the software-module Thermal Solver where the power module of one bridge leg of the power circuit is set up as a 3D-model. By pressing the '*Calc. RC-Model*'-button, the Impedance Matrix Method is applied automatically, and a fully thermal model for six semiconductors is built and stored in a file. **(b)** Dialog window for setting switching loss curves giving switching energy dependent on current for a given voltage and junction temperature. The values can be taken from measurement or directly from datasheet.

The whole procedure is quite simple and the emphasis is not on thermal modelling but still on power circuit design and control, which is the main domain of the power electronics design engineer. The right-hand side diagram in Fig.12 shows the resulting curves including transient junction temperature of one IGBT shown in magenta and one mains-side diode (red curve).

4. Multi-Disciplinary Simulation Circuit - Electromagnetic

4.1 Building Block Simulation

When designing compact power electronic building blocks (PEBB), it is crucial to take into account the parasitic electrical and electromagnetic effects of packaging and interconnection elements, and to consider them as parts of the circuit. Therefore, compact macro models of terminals, substrate layouts and bond wires of IGBT modules, as well as AC and DC bus bars connecting the module terminals to the rest of the PEBB circuitry, are needed. In recent years,

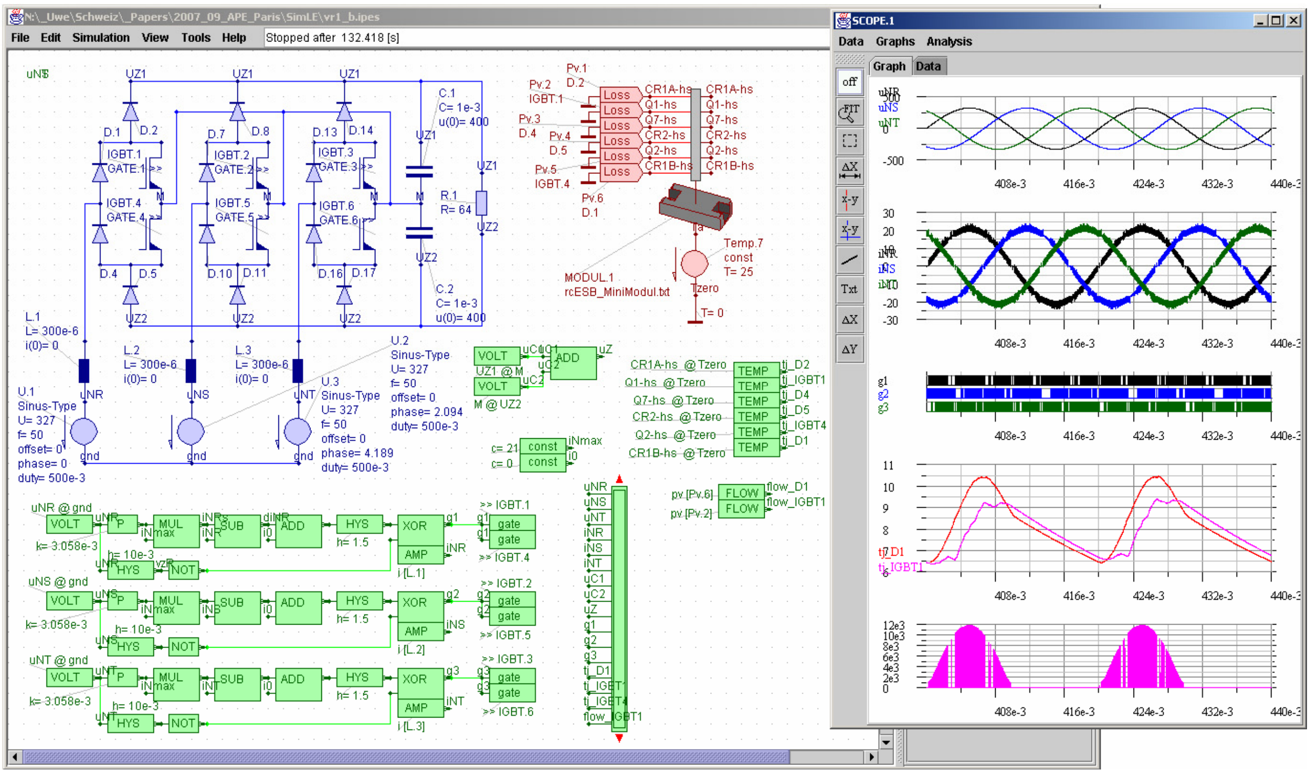


Fig.12. Screenshot from the Power Electronics Virtual Design Platform for coupled simulations circuit-thermal-electromagnetic-reliability currently under development at PES, ETH Zurich. The power circuit is shown in blue color, the control circuit in green and the thermal circuit in red. The numerical simulation over one mains period of a Vienna Rectifier (three-phase AC/DC converter, [44]) took about 2min on a Pentium 4 (3.6GHz, 1GB RAM).

many publications have shown that the PEEC (partial element equivalent circuits) method is an efficient means to extract equivalent impedance matrices of such package elements [45]-[48]. These impedance matrices can be ported to different formats for use in circuit simulators.

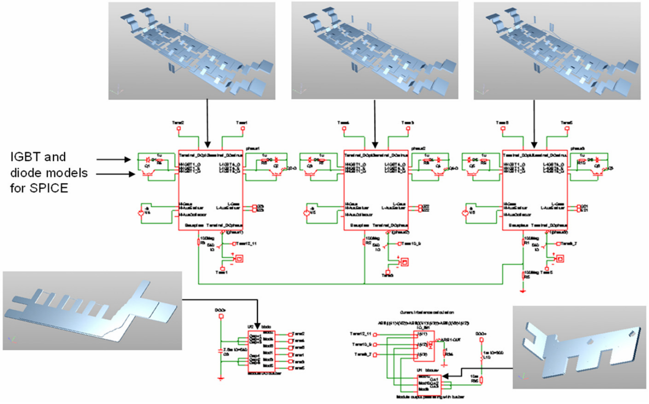


Fig.13. Top-level SPICE model with PEEC macro models of IGBT module packages, AC bus bars and DC bus bars.

Figure 13 shows the SPICE circuit of an inverter phase leg with three paralleled IGBT modules [49]. The SPICE circuit includes macro model sub-circuits for the three module packages and the AC and DC bus bars, respectively. The advantage of such detailed models becomes obvious when comparing measured and simulated turn-off transients (**Fig.14**). The curves clearly demonstrate the achievable accuracy for on-state current balancing, turn-off currents and voltage overshoot characteristics.

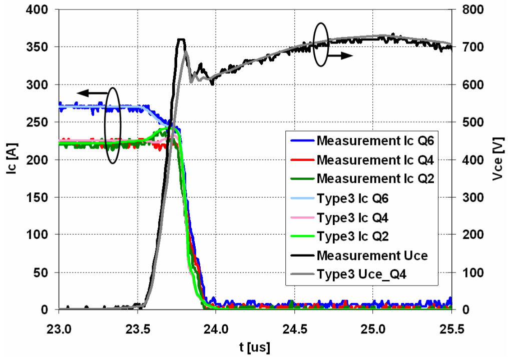


Fig.14. Comparison of measured and simulated turn-off waveforms.

4.2 IGBT Module Design Optimization

Relevant design objectives for IGBT modules are (amongst many other) high current capabilities of power terminals, low stray inductance in commutation path and uniform current and loss distribution between paralleled chips. Combining EM field calculations with circuit simulation is an efficient means to optimizing these parameters. **Figure 15** shows the computed 3D H-field pattern inside a power module caused by the high dI/dt during switching transients. The field vectors allow to identify critical chip placements and substrate layouts where strong magnetic fields might induce unwanted voltages in the gate signal loops. Transient SPICE simulations using extracted impedance matrices (**Fig.16**) and accurate IGBT and diode models show how these induced voltages affect the dynamic current sharing. In this case ([50]) the H-field pattern was computed with FLO/EMC, a TLM method based EM field simulator. The SPICE macro model was extracted using the PEEC method.

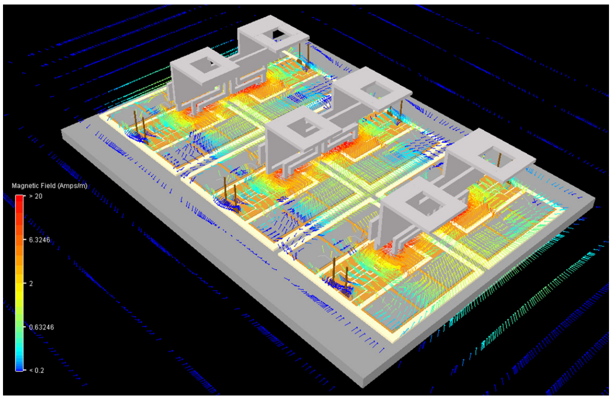


Fig.15. Magnetic field vector patterns inside IGBT module indicating potential coupling paths into gate-drive circuit.

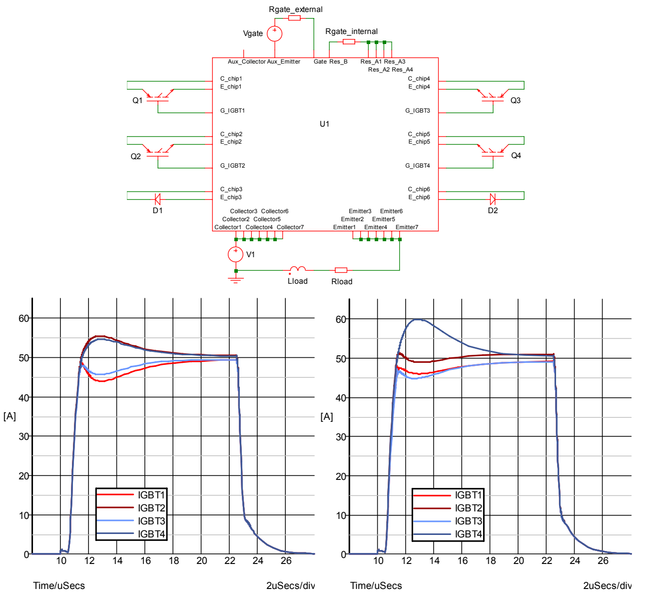


Fig.16. **(a)** SPICE circuit with module package sub-circuit. **(b)** Simulation results of dynamic current sharing between paralleled IGBTs for two different substrate layouts.

4.3 Simulating Conducted EM Noise of a Matrix Converter

The design of differential mode (DM) and common mode (CM) EMC input filters is a fundamental issue of converter design. The filter strongly influences the converter performance concerning power density or control dynamics. The filter design must ensure the compliance to EMC standards, so that the converter can operate in a non-ideal environment with external disturbances, and does not interfere with neighboring electronic equipment. The conventional design procedure is to physically build a converter system, to measure the EM noise and conducted emission levels, and to perform design iterations for the input filter until its noise measurements fulfill the required EMC standard [51]. An alternative method of designing a filter is to simulate the converter behavior. This method permits noise prediction without the need to build hardware. However, this method must include all relevant parameters in the system model, which may lead to an extraordinary complex simulation. This complexity and the lack of a combined tool for exact stray parameter extraction and circuit simulation is the reason why EMC simulation has been mainly neglected in the converter design process until now. For parasitics extraction, the PEEC method emerged as a computational effective and accurate technique. Therefore, it makes sense to adapt this technique for power electronic systems (e.g. [52]) in order to optimize PCB layouts before manufacturing. The main objective described in [53] is the evaluation of tools to predict the conducted emission (CE) levels of a complex power converter system. In the following we present a simulation of the CE of a reverse blocking IGBT based Indirect Matrix Converter (RB-IGBT IMC) [54].

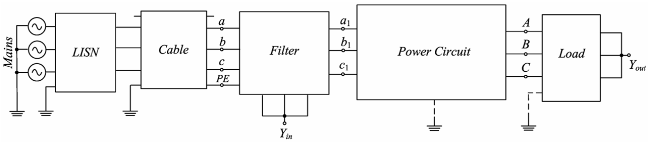


Fig.17. Block diagram of the IMC prototype and model. The DM and CM noise measurements are performed with front end Line Impedance Stabilization Network (*LISN*). The source for EMI noise is the *Power Circuit*. The DM noise propagates through the filter and the cable towards the LISN, while the CM noise is caused by step voltage changes and propagates mainly through parasitic impedances to protective earth (indicated by dashed lines).

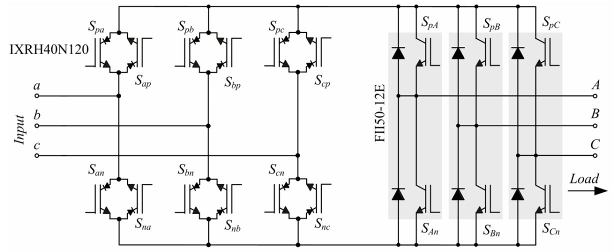


Fig.18. Schematic of the IMC power circuit.

Figure 17 shows the block diagram of the complete system, where each block is constituted of a more detailed circuit sub-model like the one shown in **Fig.18**. The load of an IMC is typically an induction motor; to keep the model more concise it was replaced by a three-phase passive *RL* load and parasitic capacitors. The converter topology shows a high complexity (18 switches, see Fig.18) and is a good demonstration object to show that EMI prediction is possible, even for complex systems. For an evaluation the converter control this modeling level of the power circuit would be adequate. However, for an EMI simulation which should be accurate up to 30 MHz, the model must include a transistor model that depicts the switching transient accurately as well as its parasitic depletion layer capacitances. In this study a first order behavioral IGBT model proposed in [55] is used and shown in **Fig.19**.

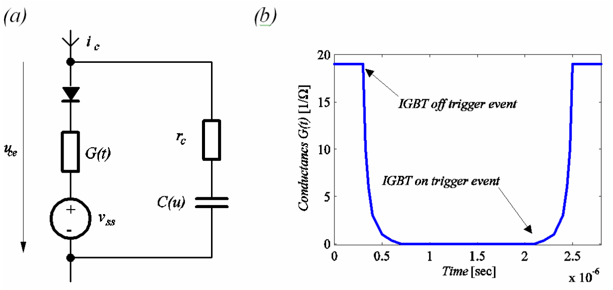


Fig.19. Behavioral IGBT model, details see [55].

The six-layer PCB layout (**Fig.20**) of the IMC contributes to the CM and DM noise via stray parasitics, namely self- and coupling inductances and capacitances. These parasitics have been extracted with a PEEC simulation which results in an impedance network that extends the power circuit in Fig.18 to a more complex simulation model. Likewise, the dominant parasitics of the other building blocks EMI filter, cables and load are included in the overall model. More details about the hardware setup and simulation can be found in [53], [54].

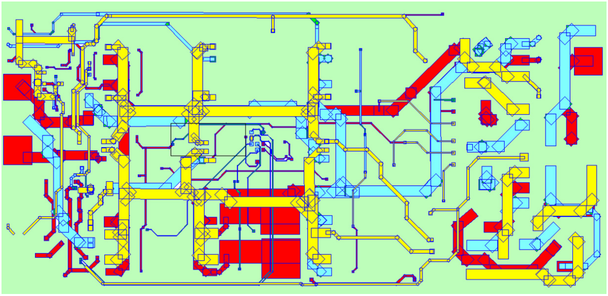


Fig.20. PEEC model of the three power PCB layers of the six-layer IMC printed circuit board.

In general one should consider a frequency domain simulation for an EMI prediction, but due to the complex control scheme of the IMC and the inclusion of the finite switching transient, this would result in severe modeling problems. Therefore, a time domain circuit simulation of a few mains periods was performed using a numerical step width of 10ns to achieve the necessary frequency resolution. The overall simulation time took several hours on a common 3 GHz personal computer. After the time-domain simulation a CE noise level spectrum is calculated by a Fourier transform of the noise voltages at the Line Impedance Stabilization Network (LISN). The simulation results and corresponding test receiver measurements are compared in **Fig.21**. The CM and DM CE levels show a very good correlation between measurements and simulations up to 5 MHz. For higher frequencies, it is probable that the non-modelled capacitive and magnetic couplings become significant.

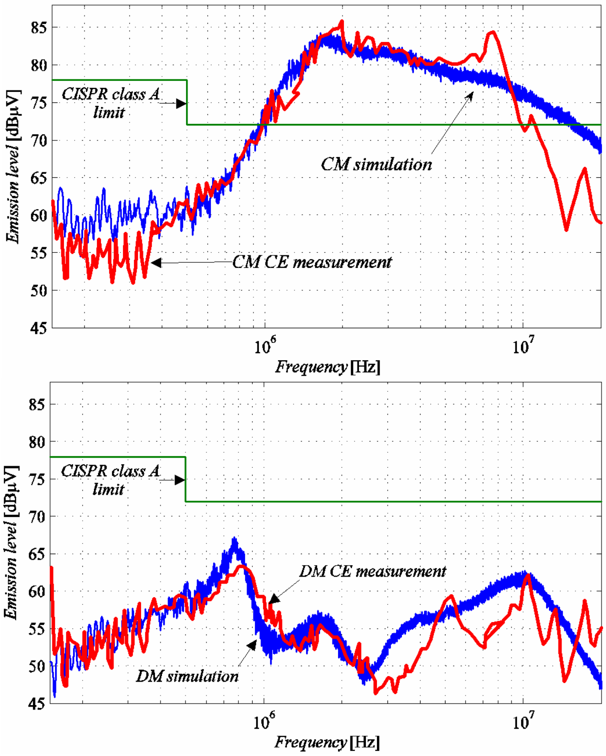


Fig.21. Differential and common mode noise measurement vs. simulation results.

This result shows that conducted EMI simulation of complex systems is basically possible with some modelling effort. A necessary precondition is the usage of an adequate parasitics extraction software, appropriate switch models and a fast circuit simulator. In the future, a tight integration between circuit simulators and an electromagnetic solver for parasitics extraction will ease the time-consuming and error-prone process of EMI model generation.

5. Conclusion

With the general demand in power electronics towards increased power density, efficiency and reliability, it will be more and more important to consider the multi-disciplinary character of power electronics in research and development. There, single-discipline software (circuit, thermal, CFD, electromagnetic, optimization) as already employed in power electronics design is not sufficient for comprehensive analysis. Accordingly, user-friendly multi-discipline simulation packages and platforms providing especially simultaneous circuit-thermal and circuit-electromagnetic coupling, and furthermore the ability to estimate reliability, and augmenting typical design tasks like EMI filter design, will become essential and could be available by 2010. In a next step such programs then would have to provide means for modelling on a system level, i.e. the interconnection and operation of several converter systems, with automatic selection of the model abstraction level as a main challenge.

References

[1] W. G. Ireson, C. F. Coombs Jr., R. Y. Moss, "*Handbook of Reliability Engineering and Management*", 2nd ed., ISBN 0-07-012750-6, McGraw-Hill, 1996.
[2] www.cadence.com/products/orcad/pspice_a_d/index.aspx
[3] www.analogy.com/products/mixedsignal/saber/saber.html
[4] www.ansoft.com/products/em/simplorer/
[5] www.simulation-research.com/sr/sr.php
[6] www.plexim.com/index.html
[7] www.powersimtech.com/
[8] www.mathworks.com/
[9] www.icepak.com/
[10] www.flomerics.com/products/flotherm/
[11] www.ansoft.com/products/hf/hfss/
[12] www.cst.com/
[13] www.ansoft.com/products/em/max3d/
[14] www.cedrat.com/
[15] www.comsol.com/
[16] H. Hofmann, "*Das elektromagnetische Feld: Theorie und grundlegende Anwendungen*"(in German), 3rd ed., ISBN 3-211-81918-5, Springer, 1986.
[17] W. H. Press, S. A. Teukolsky, W. T. Vetterling, B. P. Flannery, "*Numerical Recipes in C: The Art of Scientific Computing*", 2nd ed., ISBN 0-521-43108-5, Cambridge University Press, 1992.
[18] U. Drofenik, D. Cottet, A. Müsing, J.-M. Meyer, J. W. Kolar, "*Computationally Efficient Integration of Complex Thermal Multi-Chip Power Module Models into Circuit Simulators*", Proc. PCC'07, Nagoya, Japan, April 2-5, 2007.
[19] H. D. Baehr, K. Stephan, „*Wärme- und Stoffübertragung*" (in German), ISBN 3-540-64458-X, 3rd ed., Springer-Verlag, 1998.
[20] U. Drofenik, D. Cottet, A. Müsing, J.-M. Meyer, J. W. Kolar, "*Modelling the Thermal Coupling between Internal Power Semiconductor Dies of a Water-Cooled 3300V/1200A HiPak IGBT Module*", Proc. PCIM'07, Nuremberg, Germany, May 22-24, 2007.
[21] D. Sullivan, "*Electromagnetic Simulation using the FDTD Method*", John Wiley & Sons, Inc., 2000.
[22] J. Jin, "*The Finite Element Method in Electromagnetics*", John Wiley & Sons, NY, USA, 1993.
[23] A. E. Ruehli, "*Equivalent Circuit Models for Three-Dimensional Multiconductor Systems*", IEEE Trans. on Microwave Theory and Techniques, Vol. 22 No. 3, pp. 216-221, 1974.
[24] G. Burke, A. Poggio, "*Numerical electromagnetics code (NEC) – Method of Moments*", Research Report, Lawrence Livermore Laboratory, Livermore, CA, 1981, www.nec2.org
[25] www.fastfieldsolvers.com
[26] R. Saleh, J. White, "*Fast-Cap: A Multipole-Accelerated 3-D Capacitance Extraction Program*", IEEE Trans. on Computer-Aided Design, vol.10, no.10, pp. 1447-1459, 1991.
[27] M. Kamon, M. J. Tsuk, J. K. White, "*FastHenry: a multipole-accelerated 3-D inductance extraction program*", IEEE Trans. Microwave Theory and Techniques, vol.42, no.9, pp.1750-1758, 1994.
[28] Z. Zhu, B. Song, J. White, "*Algorithms in FastImp: A Fast and Wide-Band Impedance Extraction Program for Complicated 3D-Geometries*", IEEE Trans. Computer-Aided Design of Integrated Circuits and Systems, vol.24, no.7, 2005.
[29] A.E. Ruehli, H. Heeb, "*Circuit Models for Three-Dimensional Geometries Including Dielectrics*", IEEE Trans. Microwave Theory Technology, vol.40, no.3, pp. 1507-1516, 1992.
[30] A. Ruehli, G. Antonini, J. Esch, J. Ekman, A. Mayo, A. Orlandi, "*Nonorthogonal PEEC Formulation for Time- and Frequency-Domain EM and Circuit Modeling* ", IEEE Trans. Electromagnetic Compatibility, vol.45, no.2, pp. 167-176, 2003
[31] J. Ekman, "*Electromagnetic Modeling Using the Partial Equivalent Circuit Method*", Ph.D. thesis, EISLAB, Luleå University of Technology, Sweden, 2003.
[32] J. W. Kolar, U. Drofenik, J. Biela, M. L. Heldwein, H. Ertl, T. Friedli, S. D. Round, "*PWM Converter Power Density Barriers*", Proc. PCC'07, Nagoya, Japan, April 2-5, 2007.
[33] J. Biela, J. W. Kolar, "*Cooling Concepts for High Power Density Magnetic Devices*", Proc. PCC'07, Nagoya, Japan, April 2-5, 2007.
[34] U. Drofenik, G. Laimer, J. W. Kolar, "*Theoretical Converter Power Density Limits for Forced Convection Cooling*", Proc. PCIM'05, Nuremberg, Germany, June 7-9, pp. 608-619, 2005.
[35] M. L. Heldwein, J. W. Kolar, "*Design of Minimum Volume EMC Input Filters for an Ultra Compact Three-Phase PWM Rectifier*", to be published at COBEP'07, Sept. 30-Oct. 4, 2007.
[36] M. Ciappa, "Selected failure mechanisms of modern power modules", Microelectronics Reliability 42, pp. 653-667, 2002.
[37] E. Wolfgang, W. Gerling, U. Drofenik, "*Reliability of Power Electronic Systems*", ECPE-Tutorial, Nuremberg, Germany, April 19-20, 2007.
[38] T. Franke, G. Zaiser, J. Otto, M. Honsberg-Riedl, R. Sommer, "*Current and Temeprature Distribution in Multi-Chip Modules under Inverter Operation*", Proc. EPE'99, Lausanne, Switzerland, 1999.
[39] www.abb.com/semiconductors.
[40] U. Drofenik, J. W. Kolar, "*A General Scheme for Calculating Switching- and Conduction-Losses of Power Semiconductors in Numerical Circuit Simulations of Power Electronic Systems*", Proc. IPEC'05, Niigata, Japan, April 4-8, 2005.
[41] S. Munk-Nielsen, L. N. Tutelea, U. Jaeger, "*Simulation with Ideal Switch Models Combined with Measured Loss Data Provides a Good Estimate of Power Loss*", Conf. Record of IAS'2000, Roma, Italy, Oct. 8-12, vol.5, pp. 2915-2927, 2000.
[42] J. Chen and S. Downer, "*MOSFET Loss and Junction Temperature Calculation Model in MATLAB*", Proc. PCIM'04, Nuremberg, Germany, May 23-27, 2004.
[43] U. Drofenik, "*Embedding Thermal Modeling in Power Electronic Circuit Simulation*", ECPE Power Electronics Packaging Seminar, Baden-Dättwil, Switzerland, June 7-8, 2004.
[44] J. W. Kolar, F. C. Zach, "*A Novel Three-Phase Utility Interface Minimizing Line Current Harmonics of High-Power Telecommunications Rectifier Modules*", Record of INTELEC'04, Vancouver, Canada, Oct. 30-Nov. 3, pp. 367-374, 1994.
[45] C. Martin, J. M. Guichon, J. L. Schanen, "*Gate circuit layout optimization of power module regarding transient current imbalance*", Proc. PESC'05, Recife, Brazil, pp. 541-546, June 12-16, 2005.
[46] C. Martin, J. L. Schanen, R. J. Pasterczyk, "*Inside a power module*", Proc. IAS'04, vol.3, pp. 1519-1525, Oct. 3-7, 2004.
[47] D. Cottet, A. Hamidi, "*Numerical Comparison of Packaging Technologies for Power Electronics Modules*", Proc. PESC'05, Recife, Brazil, June 12-16, 2005.
[48] T. Wikström, T. Stiasny, M. Rahimo, D. Cottet, P. Streit, "*The corrugated p-base IGCT - a new benchmark for large area SOA scaling*", Proc. ISPSD'07, Jeju, Korea, May 27-30, 2007.
[49] D. Cottet, M. Paakkinen, "*Scalable PEEC-SPICE modelling for EMI analysis of power electronic packages and subsystems*", Proc. EPTC'06, Singapore, Dec. 6-8, 2006.
[50] D. Cottet, S. Hartmann, U. Schlapbach, "*Numerical Simulations for Electromagnetic Power Module Design*", Proc. ISPSD'06, Naples, Italy, June 4-8, 2006.
[51] M. L. Heldwein, T. Nussbaumer and J. W. Kolar, "*Differential Mode EMC Input Filter Design for Three-Phase AC-DC-AC Sparse Matrix PWM Converters*", Proc. PESC'04, pp. 284-291, 2004.
[52] P. Musznicki, J.-L. Schanen, B. Allard and P. Chrzan, "*Accurate Modeling of Layout Parasitics to Forecast EMI Emitted from a DC-DC Converter*", Proc. PESC'04, vol.1, pp. 278-283, 2004.
[53] A. Müsing, M. L. Heldwein, T. Friedli, J. W. Kolar, "*Steps Towards Prediction of Conducted Emission Levels of an RB-IGBT Indirect Matrix Converter*", Proc. PCC'07, Nagoya, Japan, pp. 1181-1188, 2007.
[54] T. Friedli, M. L. Heldwein, F. Giezendanner, J. W. Kolar, "*A High Efficiency Indirect Matrix Converter Utilizing RB-IGBTs*", Proc. PESC'96, 1199-1205, 2006.
[55] J. Tichenor, S. Sudhoff and J. Drewniak, "*Behavioral IGBT Modeling for Predicting High Frequency Effects in Motor Drives*", IEEE Trans. Power Electronics, vol.15 (2), pp. 354-360, 2000.