

# Fast Loss Predicting Prototyping Tools for Power Electronics

Peter van Duijsen<sup>1</sup> - Pavol Bauer<sup>2</sup> - Jaroslav Dudrik<sup>3</sup>

<sup>1</sup>Simulation Research, The Netherlands, [www.caspoc.com](http://www.caspoc.com)

<sup>2</sup>Delft University of Technology, The Netherlands, [www.tudelft.nl](http://www.tudelft.nl)

<sup>3</sup>TU Kosice, Slovakia,

**Abstract:** To predict losses in power semiconductors simulation can be used. The required semiconductor model can either be a detailed dynamic model where the losses are calculated from the simulated waveforms, or a fast ideal model with loss prediction.

A detailed model requires precise verified parameters and gives long simulation times. A model that can predict losses based on measurements of the switching losses of a semi-conductor improves the verification of the parameters and shortens the simulation time considerably.

In this paper the two methods are discussed and samples are given for common Power Electronic semiconductor switches.

**Keywords:** Simulation, Modeling, Power Electronics, Semiconductor Losses, Switching Losses, Loss Prediction, Reverse Recovery, Tail-current, Temperature-dependence, EMI/EMC.

## 1 Introduction

Major power semiconductor manufacturers present regularly improved new versions of their successful semiconductors. They claim that using their new semiconductors, the on state losses, switching losses and EMI/EMC are reduced. However the losses of the semiconductors are highly dependent on chip temperature and power circuit operation. In order to choose between the various semiconductors offered and to optimize the design, one needs some help in the form of a simulation tool. Using the simulation tool [1], the temperature dependent losses for a typical semiconductor in the particular power circuit can be estimated. Unlike complex semiconductor simulation models required for the first generation of circuit simulators, modern multilevel simulation tools include models that are based on manufacturer data sheet information.

In this paper an overview on the possibilities for modeling power semiconductors such as the Diode, Mosfet and IGBT is given. The paper consists of two parts. The first part describes

theoretical aspects of the modeling. The second part shows a variable speed drive application.

## 2 Modeling requirements

To simulate the losses in the semiconductors during switching, the most obvious choice for modeling would be a model that is as detailed and complex as possible and is based on a large number of parameters coming from the fabrication process. However the main goal is to predict the on and off times during switching and the thereby associated switching losses. The correct waveform during switching could be simulated using a detailed model with verified parameters under certain conditions such as:

- Correct 3D thermal model allowing the non-linear temperature distribution on the junction. Parameters are fabrication process dependent and known for the used semiconductor
- Correct 3D electromagnetic model for the surroundings of the semiconductor

Even taking into account these conditions, still one has to cope with uncertainties like:

- Varying ambient temperature
- Components from a different fabrication line or factory
- External electromagnetic influences
- Mechanical stress influencing the thermal conductivity

Taking into account these uncertainties it seems a tremendous effort to get reliable simulation results. Simulation time is not even discussed here, but it seems obvious that such a detailed analysis leads to long simulation times.

The goal for many designers is however to get some prediction on how the power electronics will perform under a wide range of conditions. For example:

- The ambient temperature varies from -20 to +40 Celsius

- Because of mechanical stress the thermal conductivity to the heat sink can over time reduce by a factor 2
- Parameters for parts coming from various manufacturers vary several percent

Also the simulation time is of concern. To perform simulation studies for a wide range of parameter variations, the simulation has to be as fast as possible. This gave rise to the development of methods that would predict losses in semiconductors during switching. There are two important design criteria that have to be predicted during the simulation:

- Switching losses
- EMI/EMC

Instead of using detailed models, manufacturers provide measurement data for switching losses, dependent on the most important conditions such as temperature and power level. EMI/EMC is mainly dependent on the electromagnetic properties of the circuit, such as lead wire inductance and bus bar inductance impedance.

### 3 Dynamic Spice based models

Spice models are the first choice when it comes to modeling semiconductors. Many manufacturers provide ready to use models and parameters for the popular spice simulators. In this section we will discuss the spice model for the diode and the mosfet.

#### *Dynamic Spice compatible diode model*

The dynamic diode model has to model the dynamic behavior of the diode. The diode has a non-linear junction capacitance that is voltage dependent. Also charge is stored in the diode during forward conduction. By turning the diode off with a negative biased current, this charge is removed from the junction. Although this process could be seen as charging and discharging the junction capacitor, the process is more complicated. The problem with the stored charge in the diode is that the charge is not stored in one p-n junction as is the case for a signal diode, but there are more regions in a power diode, due to its p-i-n structure. Therefore the non-linear capacitance model as used in Spice is not adequate enough for modeling the reverse recovery process. In Caspoc the Spice diode model is added because of compatibility, where the reverse recovery is approximated by the non-linear voltage dependent capacitor.

The standard diode model in Spice was originally developed for low power signal diodes. These

models were especially created for modeling signal diodes. Important for signal diodes is not only the V-I characteristic, but also the junction capacitance. The junction capacitance in the spice diode model is dependent on the junction voltage. This makes it possible to model varicaps, where the junction capacitance is varied by the DC offset voltage. Varicaps are used in RF designs to tune oscillators.

The disadvantage of the Spice model is the lack of an adequate reverse recovery model. However the diode spice model is fully modeled in Caspoc for compatibility reasons. The Caspoc - Spice model can be found in the library under semiconductors. In the simulation below the red trace in the scope shows the reverse recovery current of the diode. Observe the model parameters in the text editor. In this simulation the reverse recovery is modeled by the non-linear capacitor CJO.

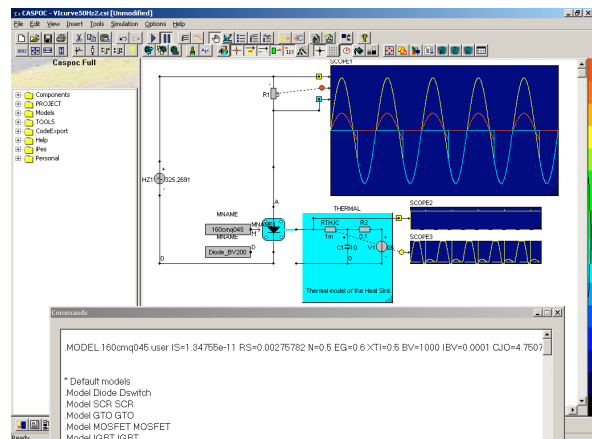


Figure 1: Spice diode simulation for a 160 Ampere 160cmq045 diode from International Rectifier.

Although the manufacturer directly provides the parameters for this model, they do not account for the influence of the forward current and slope of the turning off current.

Note the current required charging the depletion capacitance of the diode. Since a non-linear capacitor is used to model the depletion capacitance, the reverse recovery is not as sharp as with the reverse recovery model for the diode in Caspoc. The current is already flowing for negative  $V_{AK}$  due to the charging of the depletion capacitance.

#### *Dynamic Spice compatible Mosfet model*

The standard mosfet model in Spice is not suited for power mosfet simulations, since it lacks two important features:

- There is no voltage dependent behavior of the mosfet capacitors in Spice
- The Spice model is not temperature dependent, it is based only on constant temperature

In Caspoc the spice compatible model for the Mosfet is adapted where:

- In the Caspoc Mosfet model, the capacitance CGD is depending on the voltage VDG across it [2].
- In the Caspoc Mosfet model RDS(on) and KP are temperature dependent.

Figure 2 shows a simulation of single Mosfet with simple thermal model and ideal diode model for the freewheeling of the coil current

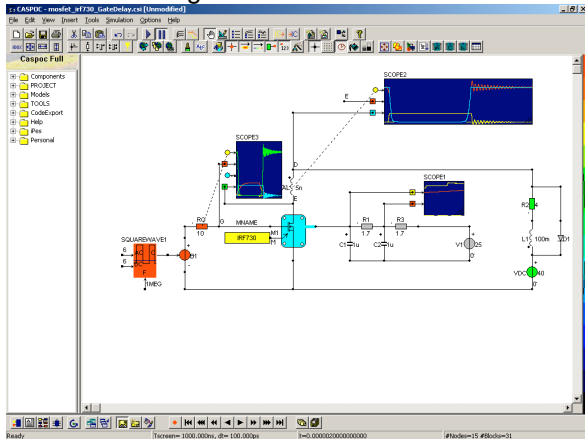


Figure 2: Spice compatible IRF730 Mosfet Model in Caspoc

Figure 3 shows the gate voltage and the relation to the turn on and turn of shown by  $V_{DS}$  and  $I_{DS}$  in case that snubber circuits are added and the diode includes a reverse recovery model.

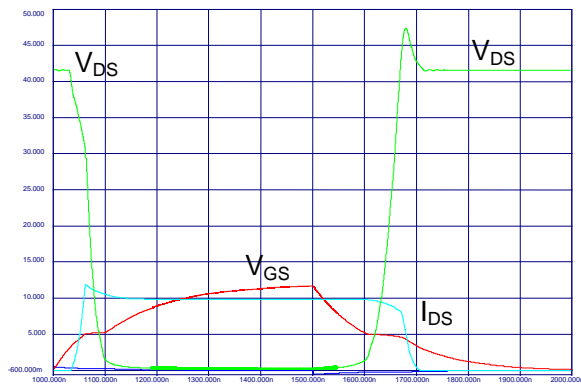


Figure 3: Mosfet Switching in Caspoc; Gate voltage  $V_{GS}$  (red trace) versus  $V_{DS}$  (green trace) and  $I_{DS}$  (blue trace)

### Spice Sub-circuit models

To overcome the problem of non-linear capacitors in Spice, manufacturers are using sub-circuits to model the mosfet. The bases of most of these models is a mosfet to model the

transconductance of the mosfet,  $I_d=f(V_{ds}, V_{gs}, V_{gs}, V_{gto})$  Extra circuitry is used to model the non-linear behavior of CGD.

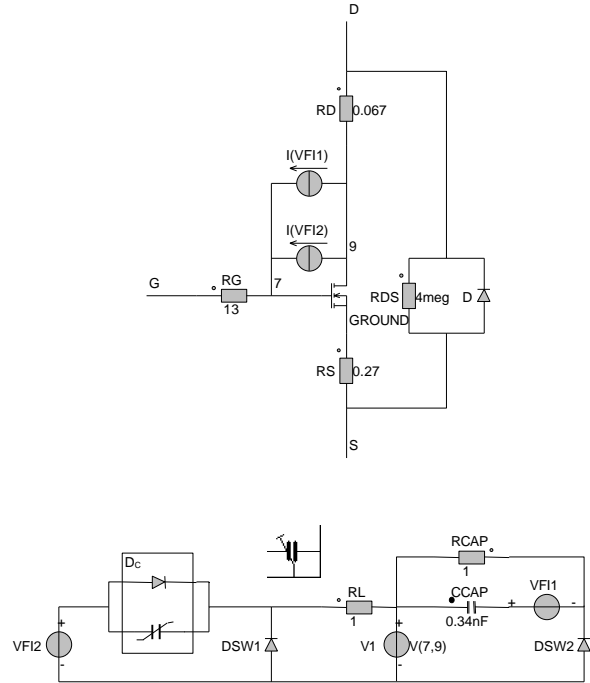


Figure 4:IR IRF530 Mosfet lumped-circuit model

The sub-circuit model for the mosfet is applied in a simulation. Because this model is optimized for a spice simulation, the spice simulator is chosen to carry out the simulation.

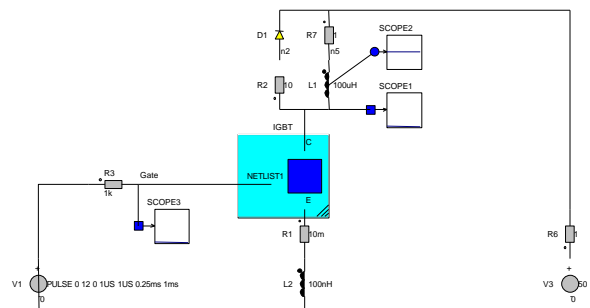


Figure 5: Spice IGBT sub-circuit used in Caspoc for export to Spice

The circuit is modeled in Caspoc where the initial conditions for the capacitor voltage and inductor current can be calculated in a system simulation in Caspoc. For the detailed simulation of the turn-on and turn-off, spice is chosen, because of the specific model specified by the manufacturer. The temperature is constant in this simulation and the build in diode model in Spice models the freewheeling diode. The manufacturer also provides the parameters for the diode.

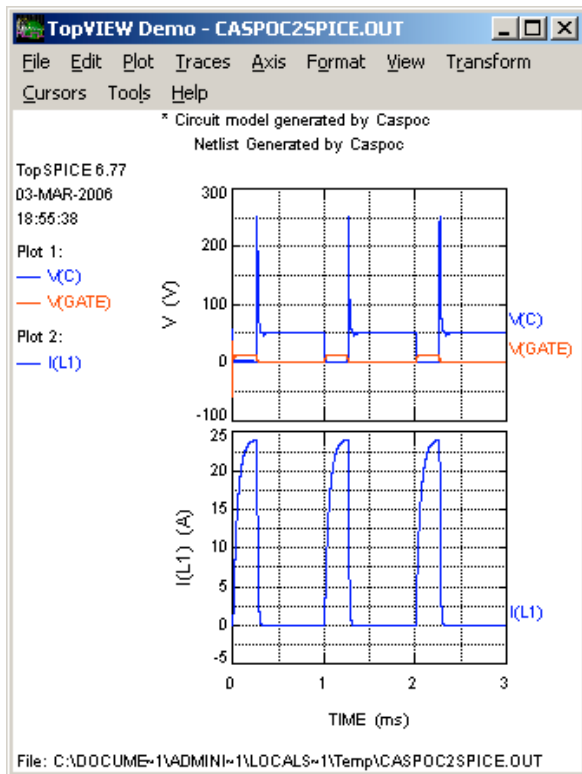


Figure 6: Results from the spice simulation.

The Spice simulator is directly called from Caspoc, so no extra modeling is required. Although various Spice models for semiconductors exist they mostly are a workaround for the shortcomings of Spice. The non-linear behavior of the capacitors is solved adequately by most manufacturers, but the temperature dependence is not always modeled. Especially for Power Mosfets the Drain-Source on-resistance and the transconductance coefficient KP is dependent on the junction temperature, which influences the static conduction losses. For more information on the Spice mosfet Sub-circuit, see [3]

#### 4 Thermal modeling

Thermal modeling is required both for the detailed models and for the fast-lost predicting models. Important is to identify the layers of material that have different thermal conductivity [4]. The ambient temperature will be nearly constant and the temperature on the heat sink will vary very slowly. However the temperature on the junction will rise much faster because of the switching losses. The example in the last section of this paper shows the temperature fluctuations directly on the junction compared to the slowly varying heat sink temperature. In the following figure the various layers of the chip,

solder, lead frame and case are shown. Thermal resistors model these layers.

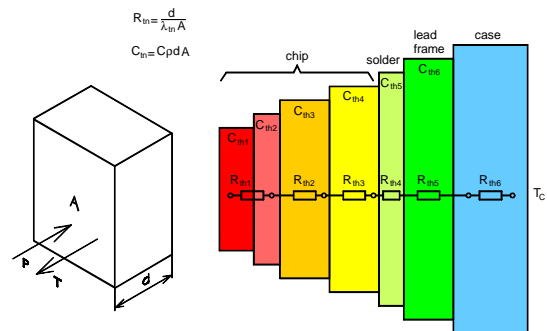


Figure 7: Layers and the thermal model of a semiconductor

The model of the semiconductor is dependent on the temperature of the junction. Therefore the losses are injected in the thermal model and the temperature of the junction is used to change the semiconductor parameters. The following figure shows the dependence of the transconductance and on-resistance of a power mosfet on the junction temperature from the thermal model.

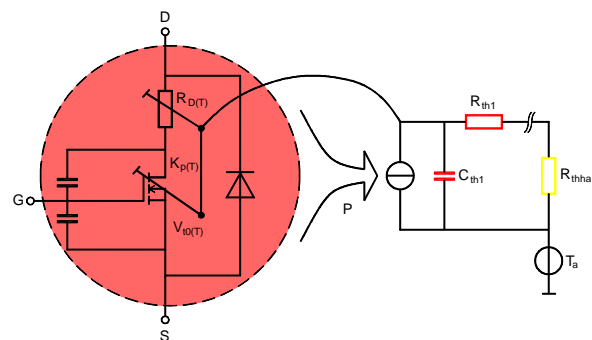


Figure 8: Coupling between the semiconductor model and the thermal model.

For simple thermal structures this model is a good approximation. However when components are spaced close to each other their thermal model will also be coupled. In that case the entire mechanical structure has to be modeled and the thermal model for this structure has to be approximated. There are two options for coupling the thermal model with the semiconductor simulation:

- Coupled simulation
- Thermal model approximation

A coupled simulation can be carried out between Caspoc [8] and Ansys/Workbench [7]. In this case the switching losses are injected in the 3D thermal model in Ansys/Workbench, where the temperature is calculated each time step. The

temperature is then send back to Caspoc to adapt the temperature dependent parameters in the semiconductors.

Another method is to first calculate a thermal model that can be included directly in the system simulation. The disadvantage here is that non-linear temperature dependent thermal conductivity parameters are neglected. In the coupled simulation, these effects are taken into account.

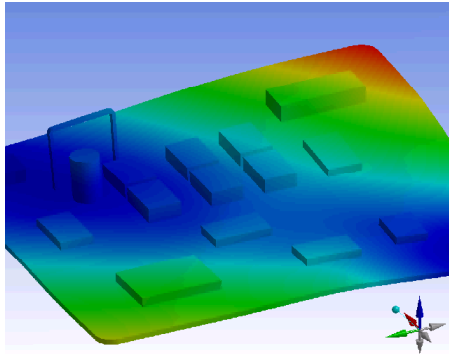


Figure 9: Thermal dependency between components on a PCB modeled in Ansys/Workbench

## 5 Fast loss predicting models

To predict the losses in a drive system, the simulation has to run for many cycles. With the ever-increasing switching frequency, the total simulation time would be too long for a simulation employing dynamic Mosfet, IGBT or GTO models. Therefore the ideal switch model, with conduction and switching losses modeled are used to calculate the losses in the inverter.

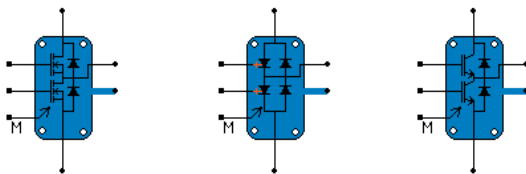


Figure 10: Modules for the Mosfet, GTO and IGBT with loss prediction

The figure above shows the modules for the fast loss prediction models. Besides the electrical nodes, each module has a thermal node that can be connected to a thermal model.

In case of a Mosfet, the transconductance and drain-source on-resistance are dependent on the temperature. In the fast loss prediction modules also a forward voltage drop for the mosfet is included in the model, although mostly the temperature dependent drain-source on-resistance is the design parameter of interest. For the IGBT, both  $V_{CEon}$  and  $R_{CEon}$  are

temperature dependent. The switching losses are given in the manufacturer data-sheet and are specified for 25° Celsius and 125° Celsius. The junction temperature has to be calculated during the simulation and is used to adapt the parameters for the semiconductors.

Using the fast loss prediction model enables the prediction of the system behavior and prediction of the losses of the component. In an IGBT, the temperature dependent  $V_{CEon}$  and  $R_{CEon}$  model the conduction losses. The switching losses are calculated from the data-sheet parameters  $E_{on}$  and  $E_{off}$ . The temperature on the heat sink is dependent on the losses. The losses are temperature dependent because  $V_{CEon}$  and  $R_{CEon}$  are temperature dependent.

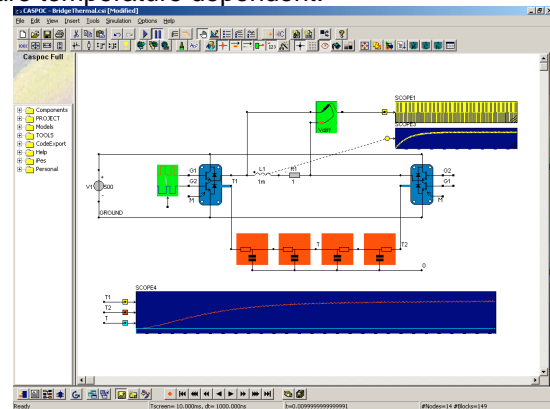


Figure 11: Single-phase IGBT inverter with thermal model

The simulation above shows the start up (first 10ms) with a switching frequency of 10kHz and runs only a couple of seconds.

The heat sink is modeled and the temperature on the case is shown and compared to the heat sink temperature. The red trace shows the junction temperature.

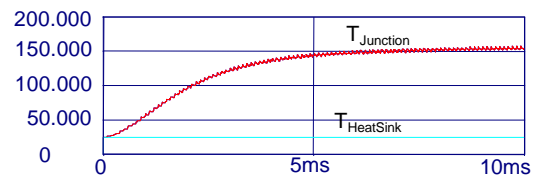


Figure 12: Junction temperature (red trace), versus heat sink temperature (blue trace)

## 6 Example of Variable Speed Drive

In this section an application is shown where the power electronics in a variable speed drive is simulated using multilevel semiconductor and thermal models.

For the rectifier we are interested in the losses and EMI/EMC that is introduced by the commutation overlap and reverse recovery.

Therefore we will use a model for the SCR and diode where the reverse recovery is modeled. The simulation results show us losses in the semiconductors and the harmonics in the input currents. From these results temperature rise in a heat sink and on the junction can be calculated and the EMI/EMC spectrum can be calculated from the input currents.

For the inverter we will use a model for the IGBT that is capable of predicting the losses and temperature rise on the junction, based on data from the data sheet of that module. Using this model we are able to model the complete inverter connected to the rectifier and DC bus, with all parasitic components included, such as ESR, current sensors and snubber-circuits. The losses in the two IGBTs and freewheeling diodes in the first inverter leg are predicted and displayed in figure 13. The temperature on the junction of the IGBT is kept constant. For varying junction temperature (25°, 75° and 125° Celsius), the losses are shown in figure 13. The start-up of the induction machine takes 1 second. The losses after the start up are clearly visible and range from 1420Watt for 25° Celsius up to 1600 Watt for 125° Celsius

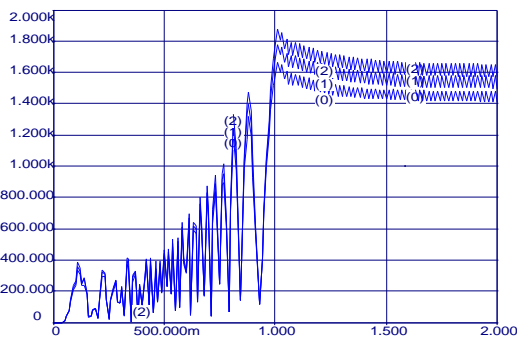


Figure 13: IGBT losses for the first phase in the inverter, for Temperature = 25°, 75° and 125° Celsius.

Figure 14 shows the start up of the induction machine for the different temperatures. The angular speed of the rotor is lower for higher temperature. Figure 15 shows the complete

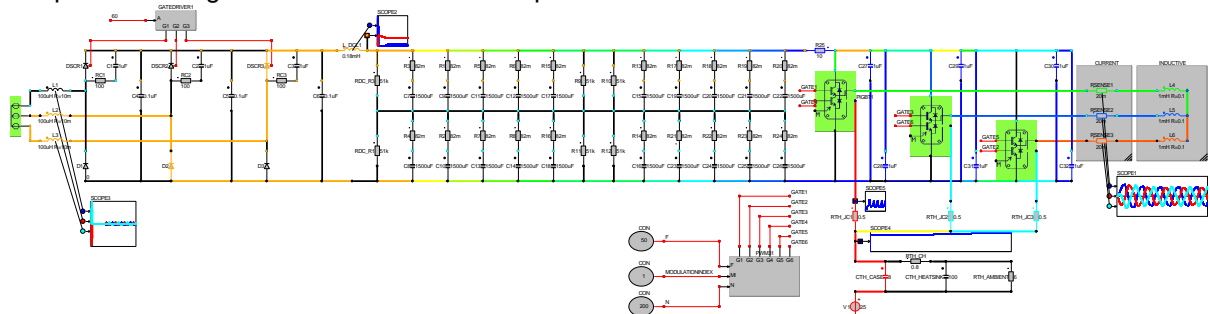


Figure 15: Variable Speed Drive in Caspoc with fast loss predicting semiconductor models and thermal simulation

simulation model in the variable speed drive. The scope shows the output currents.

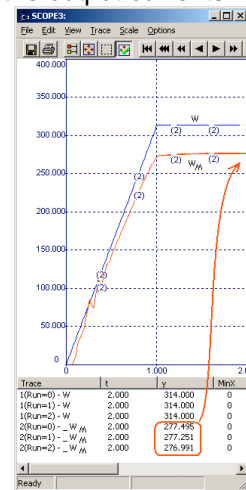


Figure 14: Reference angular speed and mechanical angular rotor speed for Temperature = 25°, 75° and 125° Celsius.

From the simulation results the temperature rise of the case and the temperature rise of the junction can be studied. As expected the temperature rise on the case is very. The temperature rise of the junction however, is faster, due to the smaller thermal time constant of the junction.

### References

- [1] Bauer, P., and Duijsen P.J. van, "Challenges and Advances in Simulation," Proceedings of PESC '2005 Conference, Recife (Brazil), 2005.
- [2] Dudrık J., "New Methods in Teaching of Power Electronics Devices", I. Journal Of Electrical And Computer Engineering, Vol. 4, No. 2, Summer-Fall 2005
- [3] Van Duijsen P.J., Bauer P., Lascu D., "Selection of Semiconductor Models in Power Electronics Simulation", Proceedings of the PCIM 2003
- [4] P.J. Van Duijsen, P. Bauer, U. Killat, "Thermal Simulation of Power Electronics", PCIM 04, Nurnberg, May 25-27, ISBN 3-928643-39-8, pp.881-886
- [5] Dudrık J., Power Semiconductor Devices and their Protection, LdV, 2001, ISBN 80-89061036
- [6] Ramshaw R.S., Power Electronics Semiconductor Switches, Chapman & Hall, 1993
- [7] Ansys/Workbench, Ansys Corp., [www.ansys.com](http://www.ansys.com)
- [8] Caspoc 2005, Simulation Research, [www.caspoc.com](http://www.caspoc.com)