OPTIMIZING

POWER SUPPLY

DESIGN

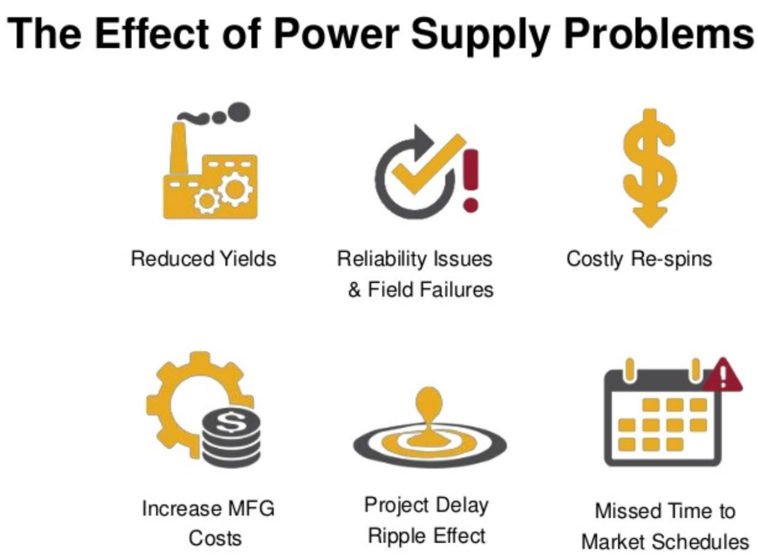
EMA Design Automation®

POWER SUPPLY DESIGN:
A ROADMAP TO SIMULATION

What is Power Supply Design?

Any electrical engineer knows providing power to your board is a key feature in PCB design. While most boards can be functional, their true quality shines when the perfect level of power to components is achieved. Building and designing better power supplies is the best way to ensure the end-product has full life-cycle potential.

But how do we ensure we can convert a (potentially variable) input voltage into a fixed output voltage? In this ebook, we will discuss important techniques to identify problems early on, rather than later in the design cycle when placing it onto the breadboard. Simulating your parts by design before building a prototype can identify problem areas and enable you to fix them before making costly mistakes.



In a larger context, consider the effects of the problems you may run into: if found too late, you will have major setbacks.

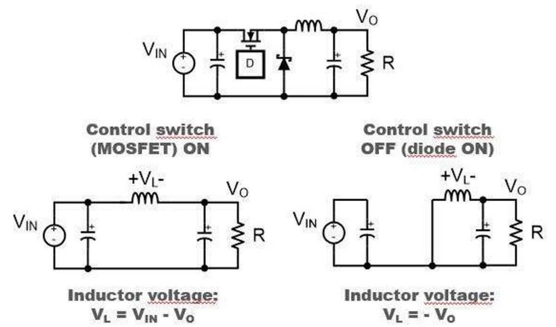
The project will be affected by reduced yields, reliability issues, costly respins, increased costs, delays, and ultimately, missed time-to-market schedules. Building a full and accurate model is key to ensuring your power supply will work as intended. Doing so allows for the capacity to peer into every possibility and adjust for problem areas. These techniques will address and help mitigate design issues down the road.

THE BUCK CONVERTER

BUILDING AN IDEAL BUCK

The most common DC-DC converter is the simple buck converter.

It takes the input voltage and uses a switching controller to step down the output voltage. One key thing to note: the average output current on the load is the same as the average inductor current. As a simple power supply analysis, the controller turns on the MOSFET and the voltage across the inductor will be the difference between the input and output voltage. When the diode is on and the MOSFET is off, the inductor voltage will be the negative polarity value of the output voltage.



$$V_O = V_{IN}\,D$$

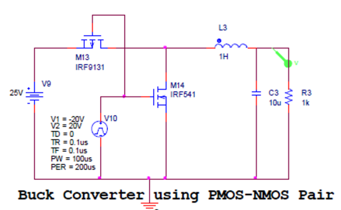
$$D = \frac{V_O}{V_{IN}}$$

Simulating your converter will help confirm you have chosen the ideal inductor and capacitor.

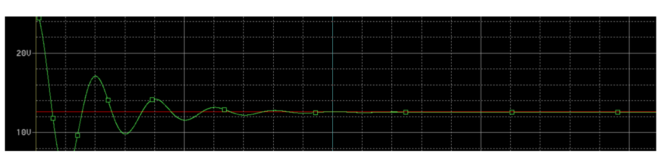
You can see in our example where placing a 25Volt input would generate a 12.5Volt output. This is an ideal, base level to begin understanding the relationships of your power supplies on a board. However, real-world examples tend to be much more complex.

- **Input voltage:** Vin=V6=25Volt
- **Output voltage:** Vo=VR1=12.5Volt

$$L_{MIN1} = D_{MAX} \times \frac{V_{IN(MAX)} - V_O}{\Delta i_{L(P-P)} \times f_{SW}}$$



Buck Converter using PMOS-NMOS Pair

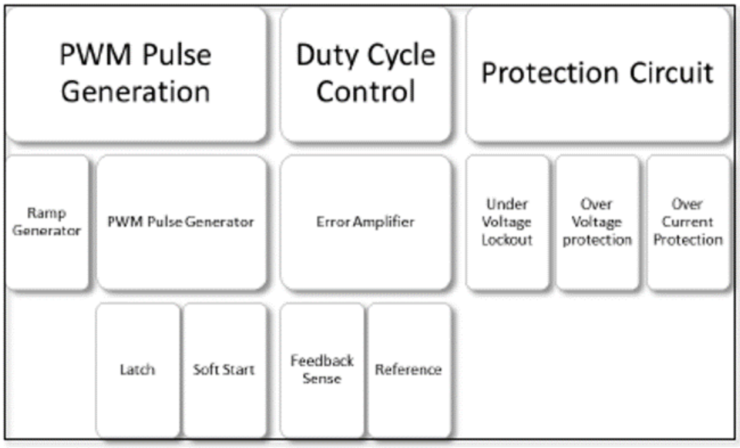


THE PWM CONTROLLER
THE MORE REALISTIC, THE BETTER

In a real-world context, you will want to find a controller model that can capture more real features.

It is difficult to find an unencrypted model of a switching controller; instead, use Spice behavioral blocks. Note that transistors are a trade-off of simulation time versus accuracy. Also, you will want to model with switches that have on/off resistances and threshold or hysteresis voltages. This is done so that you do not overheat or overload your circuit with too-high or too-low voltage pulses from events such as initial startup.

The real-world features can be found in *PWM pulse generation* or an *error amplifier,* which is tied to your feedback loop. An error amplifier is usually modeled with op-amp with feedback sense R. Other features such as *soft start, input under/over voltage protection,* and *output current limiters* can all be modeled with individual switches in a Spice circuit simulation.



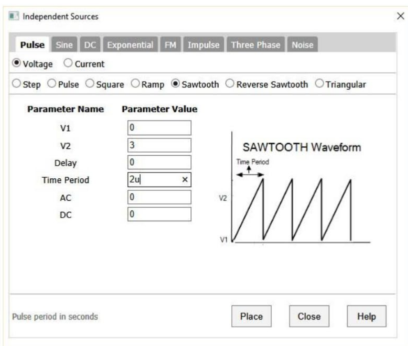
Sometimes, you may be able to find a sub-circuit model created by a manufacturer.

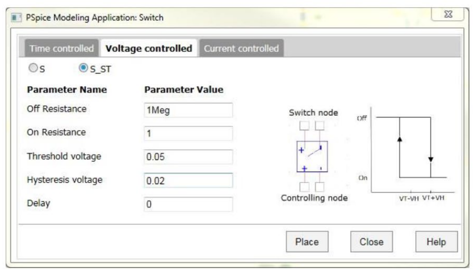
These are averaged switches that assume continuous conduction model (CCM), and are ideal not just for transient analysis, but for AC simulation and loop analysis as well. The model averages the circuit behavior to eliminate time-variance, and is a more flexible, non-linear option as a result.

PWM PULSE GENERATION,
SWITCHING CONTROLLERS, AND CURRENT LIMITERS

In the absence of an averaged switch model, there are several things you can do in your tool to replicate its features.

With your Pulse Width Modulation (PWM) controller, the pulse you output is extremely important, and can be modeled by creating a Sawtooth Ramp voltage source. Then, use a switch with some basic resistances and hysteresis voltages to mimic a comparator block for a feedback loop. You can add some delay for comparison as well.

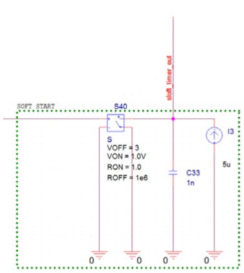


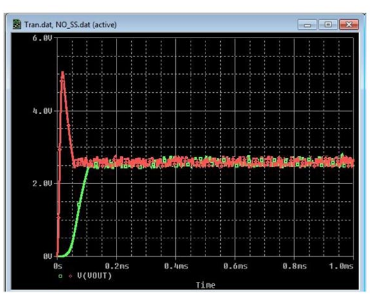


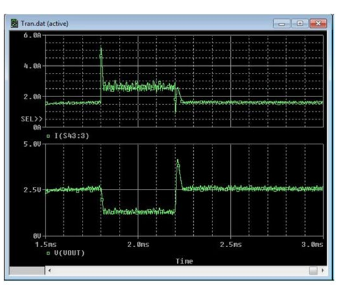
Most switching controllers will have input voltage protection, with some way to reflect controllers built onto the circuit diagram.

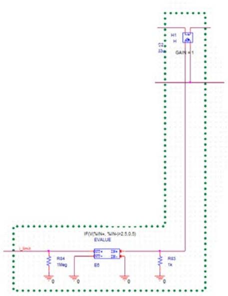
The controller's ability to turn off when the input voltage is too high or low should also be reflected in the diagram. Protecting your circuit when an input threshold is crossed is a best practice against unintentionally overloading the design. This will also extend into the use of a soft start mechanism, where a voltage-controlled switch used with a capacitor replicates a soft start function. Gradually ramping up the output voltage is ultimately more desirable, rather than creating a huge spike in output before levelling off. The soft-start function conserves power and will not cause the convergence issues that emerge from large voltage jumps.

The current limiter is also used to level off current spikes. A gain block can be used to measure the current as a voltage, then tied to a switch to trigger off a given measurement.







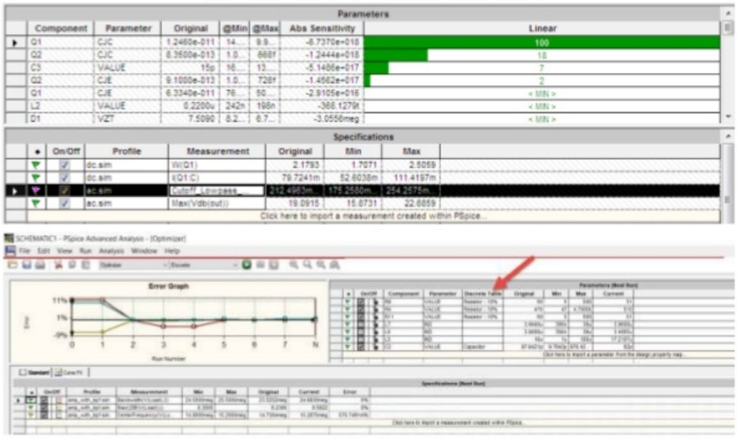


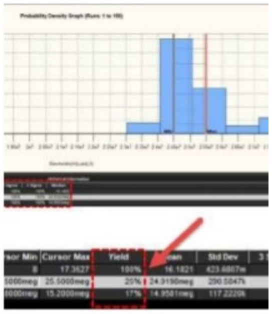
COMPONENT SELECTION

AN INPUT/OUTPUT BALANCING ACT

After modelling controller effects or finding a switch model that works for your project, you will ned to focus on the input and output ripple.

Beyond that, how the variations in component values will affect your power supply will need to be identified. The chosen inductor and capacitor components have a large effect on how the power supply is optimized. Be aware that it is not just the absolute values that matter here, but the tolerances as well. Choosing a low-quality capacitor with a too-high tolerance has the potential for disastrous results in failed power supplies.





It is important to set a desired goal for your input and output, with an acceptable tolerance level for your circuit design to perform optimally.

Once the goal is set, you can apply a *Monte Carlo technique* of random sampling in simulation to assess how variations in tolerance will affect your yield.

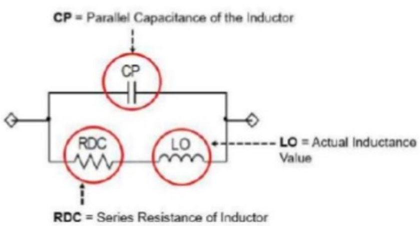
As your power supply topology becomes more complex, you can use a myriad of SPICE techniques to perform a sensitivity analysis to target key components in the design. In this way, you can optimize the design to better suit your desired goals. As a bonus, building this simulation technique allows you to easily change your design in the future when your goals mature, and new values need to be assigned.

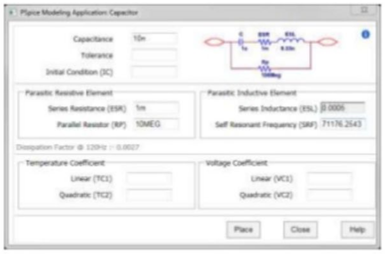
PARASITICS & MAGNETICS

'CAUSE NOBODY LIKES A POWER LEECH

Once you have chosen a general range of inductors and capacitors, you will need to simulate them in greater detail to minimize parasitics.

Parasitic capacitance is a generally unavoidable capacitance between the parts of an electronic component or circuit due to their proximity. The model should consider the equivalent series inductance and resistance (ESR) of the capacitors, as well as the DC resistance and parallel capacitance of the inductors. The self-resonant frequency of the inductors will also need to be calculated for their parasitic capacitance. Note that some capacitors (especially ceramic ones) lose capacitance as DC bias increases; this can be modeled with a voltage-controlled capacitor in SPICE. Additionally, be sure to give source and drain areas for all MOSFETs, so the junction and overlap capacitors are modeled.

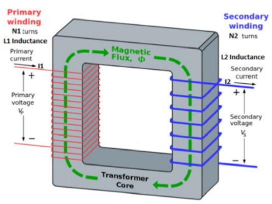


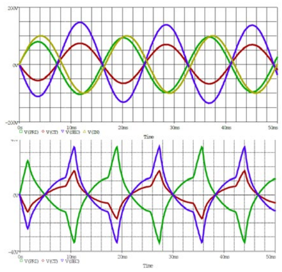


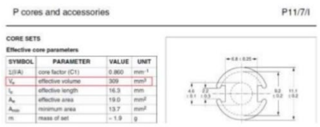
Your magnetic setup is a large factor in successful power supply design.

The ability to handle variations in the input and load will depend on it. This choice in magnetics should be an additional consideration in your inductor and transformer SPICE simulations. You will need to input the shape and core type to accurately reflect the behavior and effect on your model. A flyback converter stores magnetic energy in the inductor air gap, so without modeling the effect, the result will be inaccurate and unable to meaningfully capture issues in the design.

When magnetic parts are included, the hysteresis loss should be calculated as the core charges from positive to negative saturation and back again. This calculation is also known as a *B-H curve,* and the area inside the curve is the amount of loss. The more loss, the more heat is generated on your power supply. Once a steady state in reached in your loss calculator, you can supply the core volume of your magnetic component to calculate how much power is dissipated through hysteresis loss.





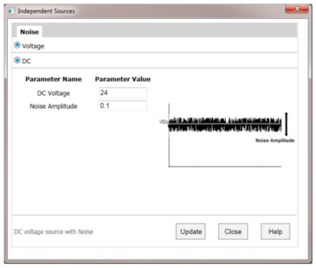


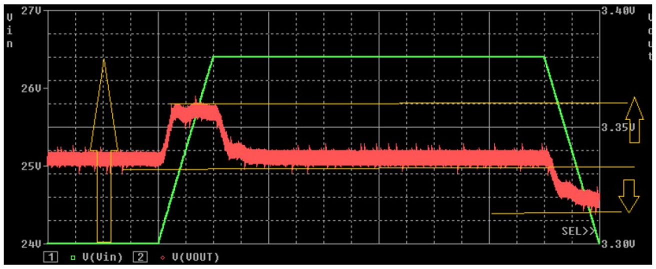
TESTING

INPUT RIPPLE & OUTPUT SHORT CIRCUITS

After generating an accurate representation of your power supply, you can start testing items such as input variation.

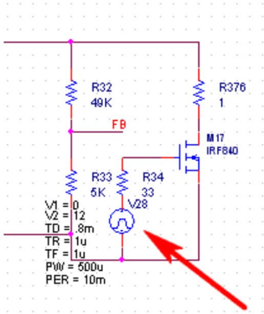
While input voltage is generally considered less important than the output voltage ripple, in many cases, having peak-to-peak noise on a previously ideal DC source or a voltage change in the input can reveal areas where your power supply design is inflexible. Another source addition can also trigger a change in input voltage to show how your circuit would respond. It is best to take the time for these testing scenarios to avoid unwanted problems down the road, when it may be too late to change them.

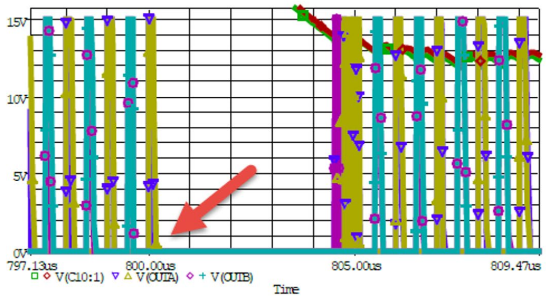




Moving on to output testing, you will want to assess how the circuit responds to load changes.

If the load is shorted, the circuit should stop supplying voltage pulses, ideally operated by the switching controller. Trigger values should be inputted to communicate when such a function should occur, then a soft start implemented to allow the voltage to level off without overloading the circuit.



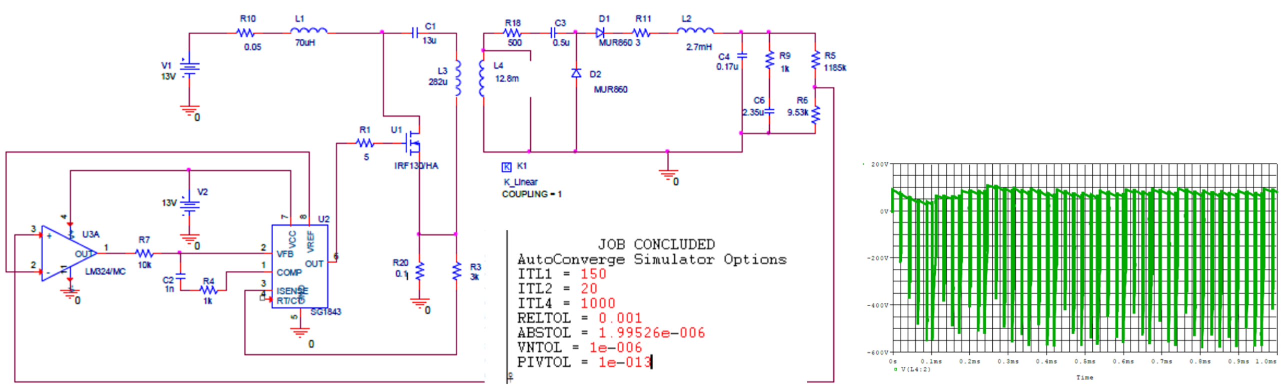


CONVERGENCE ISSUES

KNOWING YOUR TOLERANCES

In higher-frequency and higher-current power supply circuits, you may find in your simulations that convergence issues are more prevalent.

If you measure the currents running through some of your components, you will generally see they are much higher than in other simulations, so the default settings in SPICE will need to be adjusted. To avoid issues, as a rule of thumb you should ensure the absolute tolerances are not more than nine orders of magnitude smaller than the typical signals present in the circuit. Loosening the absolute tolerance (ABSTOL) or using autoconvergence when simulating power circuits will help solve the issue.



Ensuring your circuit component selections are optimized in simulation is of the utmost importance.

Any failures will occur in the models rather than later in real life when the product goes to manufacturing. Anticipating potential problems allows you to sidestep them beforehand, giving peace of mind on a reliable, working end-product. Of course, there is still one step to go before the process can advance: investigating heat generation.

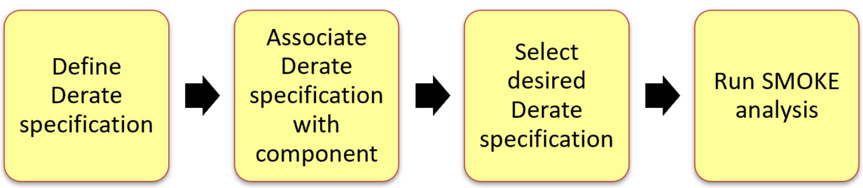
HEAT SINKS & RMS CURRENTS

TOO HOT TO HANDLE

Overheating your components is a major factor in power supply failure.

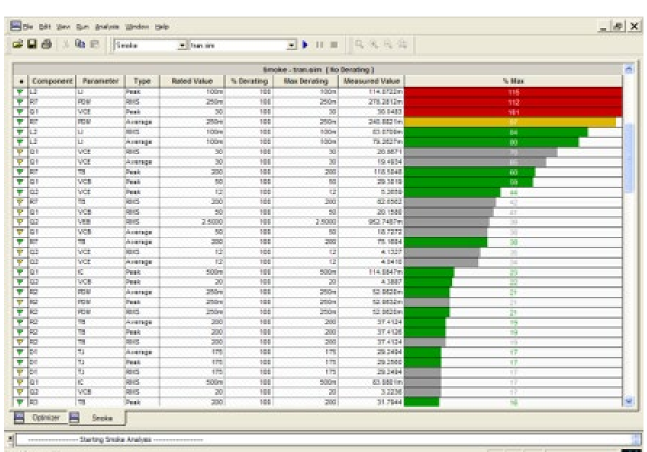
Many components are vulnerable to overheating, such as capacitors, inductors, MOSFETS, and the controller itself. Each of these components' behaviors may change in unexpected ways when the power supply heats up. Therefore, selecting an appropriate heat sink is a key aspect of any power circuit design. Simulating the heat sink will show its effect on component behavior, as well as reveal any parasitic inductance it places on the design.

There are several steps to selecting and simulating a heat sink. First, you will need to define the temperature derating specification. Then, associate the derate specification with the appropriate component and select the desired specification. Running a SMOKE analysis will detect which components may suffer the most from the heat in the power supply so you can adjust values accordingly.



RMS measurements are the best and most common way to check power dissipation (and thus heat).

In real life, it is often a challenge to measure RMS (Root Mean Squared) currents accurately on each device, so you will want to know how much the rating is exceeded for each one. In a SPICE simulation, the values are more ideal and can be easier to calculate.

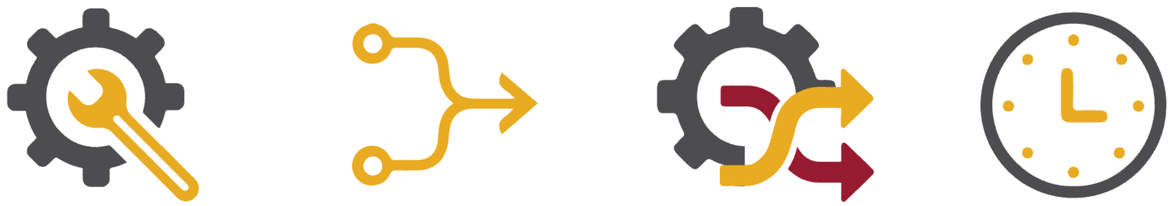


FINAL CONSIDERATIONS

YOU'VE GOT THE POWER

On a larger scale, more PCB designers are utilizing digital power supply techniques to build their designs.

When simulating in this way, you must consider analog-to-digital converters that will also need to be modeled to get an accurate result in circuit behavior. If your PWM controller is modeled in C or MATLAB, it can be a challenge to simulate along a SPICE circuit, so having the proper design tools to handle the conversion is important to your productivity.



See & Fix Violations In Real-Time | Streamline Design Release Process | Eliminate Multiple Validation Iterations | Improve Time To Market

Overall, good power supply design will enable you toward your goals of project success.

Elevating your methods allows your team to see and fix violations in real time, eliminate multiples iterations, streamline the design process, and improve your time-to-market schedule. Simulating the design and capturing real-world problems like parasitics, losses, and heat generation empowers you to tackle design changes up-front, reducing respins and lost time. Be proactive in your simulation and build better products faster with rapid prototyping abilities in OrCAD PSpice Advanced Analysis.

See more info on OrCAD PSpice Advanced Analysis, or contact us at info@ema-eda.com.

EMA Design Automation®