AN5650

VIPower transient thermal analysis using SPICE

Introduction

Calculating transient thermal response in a high side driver can get very complicated. To be accurate, transient thermal analysis should include conduction losses, switching losses, supply current losses as well as clamping energy at turn off.

ST provides a solution using a thermal analysis tool called TwisterSIM that can be downloaded from ST web page (TwisterSIM). TwisterSIM is very useful to determine the dynamic thermal response to transients from inrush to stall including clamping energy capability. TwisterSIM also provides first level selection guide based on some of application requirements. Please refer to UM1874 for details on how to use TwisterSIM.

TwisterSIM works well for short duration analyses. Durations of more than a few seconds, however, can take enormous amounts of time to process. In this case ST provides a Foster model in every VIPower high side driver datasheet that is similar to what is used in TwisterSIM. With this model, longer duration dynamic thermal analyses can be performed in SPICE.

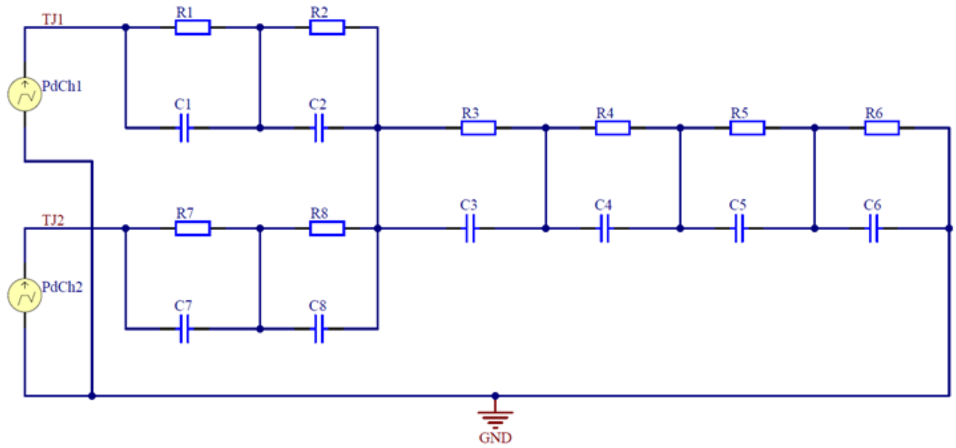
The effort comes in modelling the power dissipation. Conduction losses are more than a simple square of the current multiplied by the on-resistance ($R_{DS(on)}$) of the switch. The $R_{DS(on)}$ changes with temperature and doubles between 25 °C and 150 °C. This change in on-resistance ($R_{DS(on)}$) dramatically affects the power dissipated in the switch. As a result, a few elements need to be added to the basic Foster model provided in every VIPower datasheet.

This application note describes the process of adding the power dissipation elements to make an accurate long-term thermal analysis. This is not intended to analyze shorted loads or to emulate the functionality of the high side driver (that are thermal intervention, current limit, or inductive clamping), but to provide a thermal response to various complex loads not easily simulated in TwisterSIM. For the M07 high side driver protection strategy please refer to AN5368.

1 The Foster thermal impedance model

In every VIPower M07 high side driver datasheet there is a six order Foster model for thermal impedance. The Foster model includes provisions for each the outputs. Each model is created by curve fitting from actual thermal testing results. That means the elements do not necessarily fit the natural boundaries of an IC (die, package, circuit board). There is some correlation, however it is not intentional. The first few elements tend to be die related just due to their time constants. The last few elements are more related to the circuit board thermal capability.

Figure 1. Six order Foster model equivalent circuit for a 2-channel HSD



When using the above model in SPICE, the current represents the power dissipation and voltage represents the temperature. The channel power dissipation components can be simulated using or manipulating various current sources available on SPICE. The equivalent resistors and capacitors are adjusted based on the circuit board heat sinking area. The values for these components also vary depending on the device package and die size. The parameters used to fill in the Foster model are found in all M07 VIPower high side driver datasheets (see Table 1).

Table 1. Example of a thermal parameter table (VND7050AJ)

Area/island (cm^2)	Footprint	2	8	4L
R1 = R7 (°C/W)	1.8			
R2 = R8 (°C/W)	3.2			
R3 (°C/W)	8	8	8	6
R4 (°C/W)	14	6	6	4
R5 (°C/W)	30	20	10	3
R6 (°C/W)	26	20	18	7
C1 = C7 (Ws/°C)	0.00035			
C2 = C8 (Ws/°C)	0.005			
C3 (Ws/°C)	0.05			
C4 (Ws/°C)	0.2	0.3	0.3	0.4
C5 (Ws/°C)	0.4	1	1	4
C6 (Ws/°C)	3	5	7	18

This basic model provides for the thermal impedances based on four different circuit board layouts. It is possible choose from one of these four layouts or estimate something in between based on specific layout.

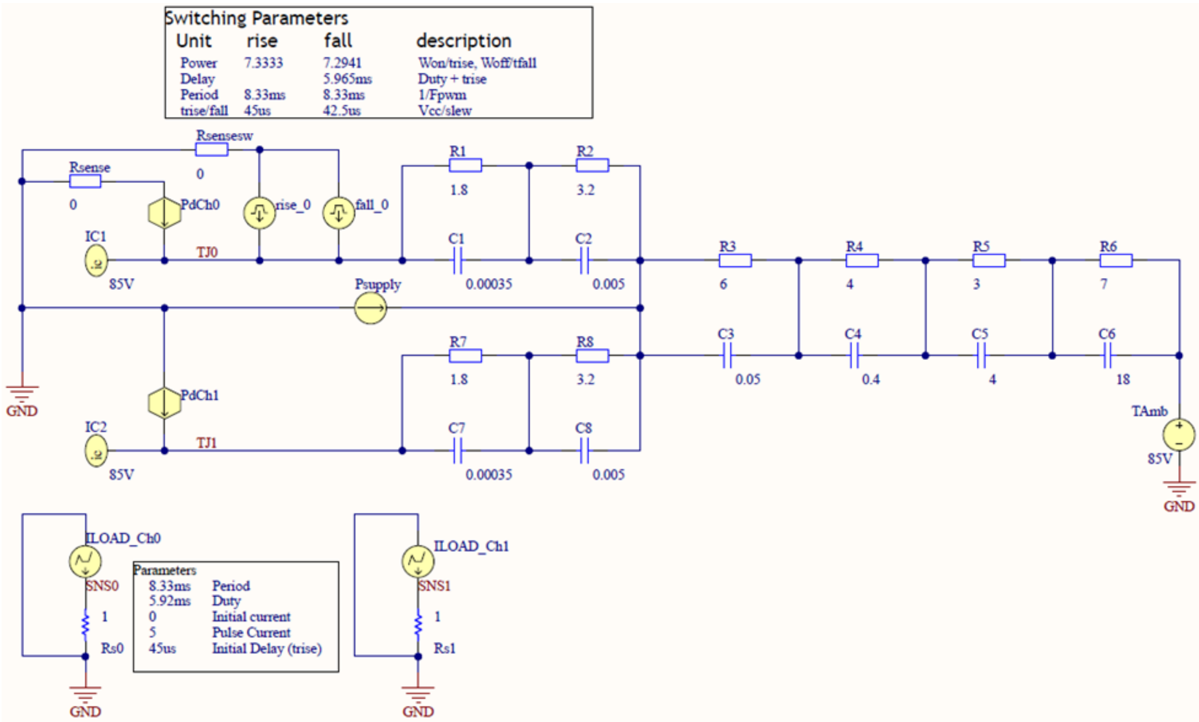
2 Modelling power elements

SPICE simulators can easily incorporate the Foster models found in our datasheets. A few elements need to be added for this model to provide accurate junction temperatures. This can be broken down into their separate contributions.

- Conduction losses
- Switching losses
- Supply current losses

The power losses can be modelled in many ways. The below schematic illustrates two different ways of modelling the power losses due to load current. The current and power dissipation sources discussed here are just two ways available. SPICE is capable to simulate several combinations of power losses.

Figure 2. Foster thermal model including power sources



2.1 Initial conditions

Initial conditions are essentially the ambient temperature at the start of an evaluation. These are defined by the ICx control statements and the TAmb voltage source in the schematic in Figure 2. The initial conditions are mandatory in ensuring the proper thermal analysis. Typically, the initial conditions and the ambient temperature (represented as a voltage) are the same. In the schematic example of Figure 2, the ambient temperature and initial conditions are 85 V.

2.2 Conduction losses

The on-resistance ($R_{DS(on)}$) of the switch is dependent on the junction temperature. To compensate for this dependency, a two stage simulation is implemented. This replaces the PdChx piecewise linear estimation of power found in the datasheet model with a two-stage system where the current is first defined and then the power dissipation can be dynamically calculated using that defined current with the junction temperature (TJx).

In the schematic shown in Figure 2, ILOAD_Chx is used to define the actual load current. Then the second stage, PdCHx, calculates the resulting power using an equation that includes the temperature effects on $R_{DS(on)}$.

2.2.1 Load current definition

- Periodic (PWM) outputs.

  Load current can be defined in any way that is convenient for the waveform. In the above example, ILOAD_Ch1 is a pulse width modulated signal set with a 71% duty cycle at 120 Hz. Because this is a periodic signal with some frequency, switching losses are also considered as another power contributor.

  This output power then has two contributors, conduction and switching. These are mutually exclusive contributors. While the driver is switching, the conduction losses are not considered and vice versa.
- Non periodic outputs.

  Non-periodic or odd duration waveforms can be described using piecewise linear (IPWL) function in SPICE. ILOAD_Ch2 for example, is a simple piecewise linear step function. Because it is not a PWMmed output switching losses do not need to be considered. A motor inrush, run, and stall currents can be emulated in this manner.

2.2.2 Calculating the conduction losses

The second stage takes the predefined current and converts it to power using the junction temperature to adjust the $R_{DS(on)}$. A reasonable approximation of on-resistance for worst case analysis doubles the $R_{DS(on)}$ between 25 °C and 150 °C. The Equation 1 provides a simple linear interpolation reflecting the thermal characteristics of on-resistance over temperature. This is not exact. However, it is accurate enough for worst case calculation purposes.

Equation 1 – Junction temperature compensated $R_{DS(on)}$

$$R_{DS(on)} = R_{DS(on)}\ @\ 25\ ºC\left[1 + \frac{T_J - 25\ ºC}{125\ ºC}\right] \tag{1}$$

This can be simplified to be more easily implemented in SPICE.

Equation 2 – Junction temperature compensated $R_{DS(on)}$ simplified

$$R_{DS(on)}\left(T_J\right) = R_{DS(on)}\ @\ 25\ ºC\left(0.8 + 0.008T_J\right) \tag{2}$$

The simple I^2R power equation for conduction losses then looks like:

Equation 3 – Conduction losses with respect to temperature

$$P_{COND} = {I_{OUT}}^2 \times R_{DS(on)}\ @\ 25\ ºC\left(0.8 + 0.008T_J\right) \tag{3}$$

What this looks like in SPICE for OUT0 using a 50 mΩ switch (VND7050AJ):

Equation 4 – Current to power equation estimation for SPICE model

$$BPdCh0\ 0\ TJ0\ I = V(SNS0) \wedge 2 * 0.05 * (0.8 + 0.008 * V(TJ0)) \tag{4}$$

Where:

- BPdCh0 is calculated power dissipation due to conduction losses in Ch0
- V(SNS0) is the voltage seen at the SNS0 node (ILOAD_Ch0 * 1 Ω)
- 0.05 is the output on-resistance at 25 °C for Ch0 ($R_{DS(on)}$ @ 25 ºC = 50 mΩ).
- V(TJ0) is the measured junction temperature at the Ch0 node (TJ0).

3 Switching losses

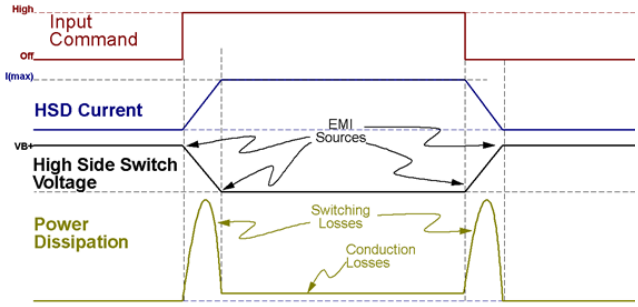
Switching losses are a result of the driver behaving as a linear output for very short periods of time. The output is neither on nor off. It is being driven linearly from one rail to the other. As a result, switching losses are calculated using voltage and current (as opposed to I^2R).

Equation 5 – Basic power equation during switching

$$P_{Switch}(t) = V_{OUT}(t) \times I_{OUT}(t) \quad (5)$$

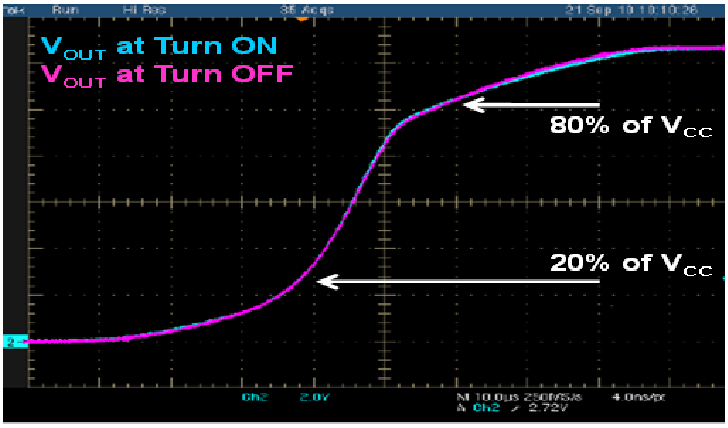
The Figure 4 illustrates the switching losses in a simple trapezoidal output. With this simple illustration, we can see that EMI comes from the "corners" of the waveform and the power dissipation is mostly in the middle of the transition. To reduce switching losses the slew rate needs to be fast. However, to reduce EMI the opposite is true.

Figure 3. **Resistive load switching losses waveform**



To reduce EMI while reducing switching losses, ST's VIPower high side drivers have rather sophisticated turn on/off slopes. These are purposely done to reduce EMI while keeping switching losses to a minimum. This makes trapezoidal estimation of switching losses rather complicated.

Figure 4. **Typical VIPower M07 high side driver switching waveforms**



With that, switching losses for VIPower high side drivers are expressed in terms of specification parameters, W_{on} and W_{off}. These parameters define the energy consumed during a single switch event (rise or fall) at a specific load and supply voltage. The advantage is a much simpler method of calculating an accurate value for switching losses.

Table 2. Switching losses definition (VND7050AJ)

VCC = 13 V, -40 °C < TJ < 150 °C, unless otherwise specified						
Symbol	**Parameter**	**Test conditions**	**Min.**	**Typ.**	**Max.**	**Unit**
W_{on}	Switching energy losses at turn-on (t_{won})	R_L = 6.5 Ω	-	0.25	0.33	mJ
W_{off}	Switching energy losses at turn-off (t_{woff})	R_L = 6.5 Ω	-	0.23	0.31	mJ

Switching losses can be calculated in a number of ways using the parameters of Table 2. The simplest is calculating the average DC power loss and adding that to the junction temperature (TJx) node. For a more detailed examination of switching losses two pulse current sources can be used, one for rise time and one for fall time. Again, these pulse current sources would be inserting current into the TJx nodes (reference the TJ0 node in Figure 2). Switching losses affect junction temperature and as a result affect $R_{DS(on)}$. However, $R_{DS(on)}$ does not affect switching losses. Therefore, contributions to switching losses are not inserted as a part of the conduction losses calculation.

3.1 Switching losses as a DC average

The simplest way to insert switching losses is incorporating a DC estimation of switching losses. This is more than sufficient for most thermal evaluations. It is by far the simplest to calculate. To calculate the losses due to switching requires multiplying the sum of the two switching parameters (W_{on} and W_{off}) by the switching frequency. This however is only accurate at the voltage and load specified in the datasheet.

Equation 6 – Switching losses using W_{on} and W_{off}

$$P_{Switch} = \left(W_{on} + W_{off}\right) freq \tag{6}$$

Switching losses, in however complex the waveform might be, are still essentially voltage multiplied by current. With that we can “adjust” the losses due to switching by swapping out the specified voltage and load for the application voltage and load. The VIPower M07 specifications use a load resistance to specify the W_{on} / W_{off} losses. The typical load definition is in terms of current. As a result, the following equation can be used to calculate the average switching losses (for a resistive load).

Equation 7 – Resistive switching losses using W_{on} and W_{off} adjusting for application conditions

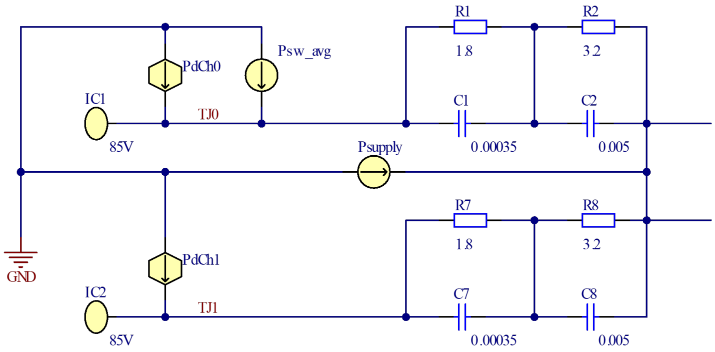
$$P_{Sw\_avg} = \frac{I_{new} V_{new} R_L}{{V_{spec}}^2} \left(W_{on} + W_{off}\right) freq \tag{7}$$

Where:

- P_{Sw_avg} are the adjusted resistive load switching losses
- V_{new} is the application supply voltage
- I_{new} is the application load current
- R_L is the load resistance used to specify the W_{on}/W_{off} parameters
- V_{spec} is the supply voltage used to specify the W_{on}/W_{off} parameters (13 V)

With this calculation the average power dissipated due to switching losses can be inserted into the TJx node as shown in Figure 5. This does not generate in the simulation results the “bumps” in junction temperature due to switching. It does, however, provide a reasonable job of representing the elevation of the average junction temperature over time.

Figure 5. Inserting average switching power dissipation

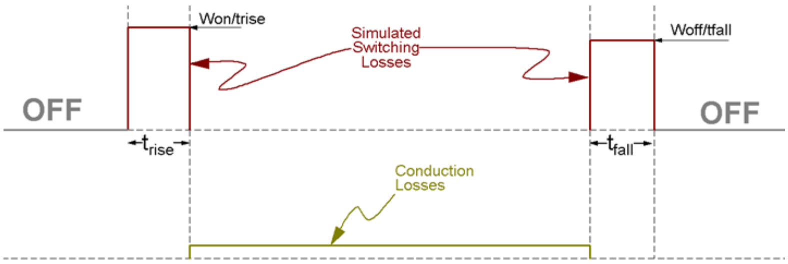


Inductive loads with a freewheeling diode generate a bit more power when switching. Due to the geometry of the waveforms, inductive load switching losses can use the same switching losses Equation 7 multiplied by 3. However, VIPower M07 high side drivers tend to switch too slowly for the switching frequency needed for most inductive loads. A quick switching losses calculation can easily illustrate this.

3.2 Instantaneous switching losses simulation

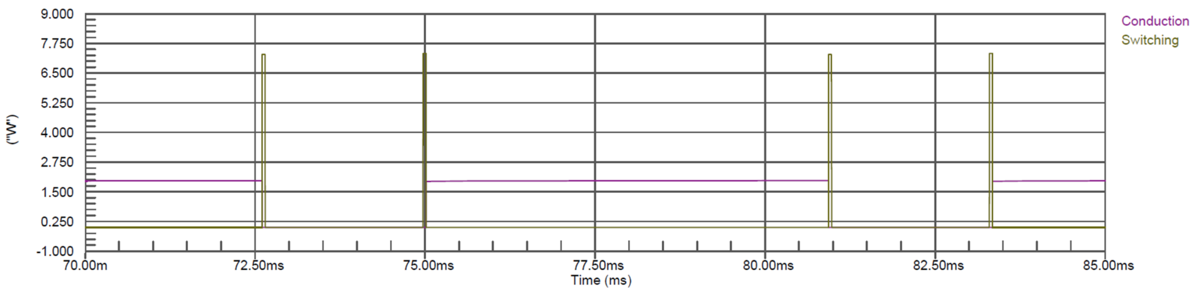
To calculate the instantaneous switching losses two pulse current sources are used. One defines the rise, or turn-on, losses and one defines the fall, or turn-off, losses. They are different for two reasons. First, they are not the same amplitude. Second, unless the duty cycle is 50%, they occur at different places in the period.

Figure 6. Simulated switching losses



Since switching losses and conduction losses are mutually exclusive care must be taken to ensure that they do not overlap. This means offsetting the pulses to sit just prior to turn-on and just after turn-off.

Figure 7. Switching and conduction losses simulation results



Offsets take into account the rise time parameter duration. Each pulsed waveform has a rise time, a duration, and a fall time. These are additive. Setting the waveform rise and fall times to zero eliminates any possible thermal contribution due to overlap. It is important to remember that this is a simulation where power sources are super positioned so that they represent actual power being dissipated. Realistic rise and fall times are not required. Only the final, superposed result is relevant.

3.2.1 Calculating the output rise and fall times

Rise and fall times are not provided in the VIPower high side driver datasheets. ST provides the expected slew rates. From these parameters it can derive the expected rise and fall times with a simple equation.

Table 3. Switching parameters for the VND7050AJ

VCC = 13 V, -40 °C < TJ < 150 °C, unless otherwise specified						
Symbol	**Parameter**	**Test conditions**	**Min.**	**Typ.**	**Max.**	**Unit**
$t_{d(on)}$	Turn-on delay time at T_J = 25 °C	R_L = 6.5 Ω	10	60	120	µs
$t_{d(off)}$	Turn-off delay time at T_J = 25 °C		10	40	10	µs
$(dV_{OUT}/dt)_{on}$	Turn-on voltage slope at T_J = 25 °C	R_L = 6.5 Ω	0.1	0.3	0.7	V/µs
$(dV_{OUT}/dt)_{off}$	Turn-off voltage slope at T_J = 25 °C		0.1	0.32	0.7	V/µs

The rise and fall time durations can be calculated using the above parameters and the application supply voltage:

Equation 8 – Rise time calculation

$$t_{rise} = \frac{V_{supply}}{\left(\frac{dV_{OUT}}{dt}\right)_{on}} \quad (8)$$

Equation 9 – Fall time calculation

$$t_{fall} = \frac{V_{supply}}{\left(\frac{dV_{OUT}}{dt}\right)_{off}} \quad (9)$$

Where :

- t_{rise} is the duration used for the rise time in the simulation (reference from Figure 2: rise_0)
- t_{fall} is the duration used for the fall time in the simulation (reference from Figure 2: fall_0)
- V_{supply} is the supply voltage of the application
- $(dV_{OUT}/dt)_{on}$ is the rise time slew rate given in the datasheet
- $(dV_{OUT}/dt)_{off}$ is the fall time slew rate given in the datasheet

The slew rate parameters in the datasheet include the minimum, typical and maximum values. For simplicity the typical value can be used. The energy is the same and the overall junction temperature is the same as the energy dissipated is the same. The only difference is a slight peak in junction temperature when using the faster slew rates. This is because there is the same amount of energy being dissipated over a shorter duration.

3.2.2 Calculating the energy in each transition

As mentioned in Section 3.1 , the W_{on} and W_{off} parameters are only valid at 13 V at the load resistance specified. As a result, some adjustment need to be done to accommodate for different supply voltages and loads. Secondly, it needs to spread the calculated power over the switching duration. It is not ideal in that the simulation (a square pulse) does not reflect perfectly the actual instantaneous power during the transition (somewhat sinusoidal). However, it is close enough for this purpose.

By adapting Equation 7 the supply and load differences for each transition can be calculated:

Equation 10 – Adjusting W_{on} for supply and load

$$E_{Sw\_on} = \frac{I_{new} V_{new} R_L}{V_{spec}^{\ 2}} W_{on} \quad (10)$$

Equation 11 – Adjusting W_{off} for supply and load

$$E_{Sw\_off} = \frac{I_{new} V_{new} R_L}{{V_{spec}}^2} W_{off} \tag{11}$$

Turning these into equivalent currents for the simulation requires to divide the result by the transition times and obtain Joules per second (J/s = W) for the duration of the switch.

Equation 12 – Calculating power during switching (rise time)

$$P_{Sw\_on} = \frac{E_{Sw\_on}}{t_{rise}} = \frac{I_{new} R_L \left(\frac{dV_{OUT}}{dt}\right)_{on}}{{V_{spec}}^2} W_{on} \tag{12}$$

Equation 13 – Calculating power during switching (fall time)

$$P_{Sw\_off} = \frac{E_{Sw\_off}}{t_{fall}} = \frac{I_{new} R_L \left(\frac{dV_{OUT}}{dt}\right)_{off}}{{V_{spec}}^2} W_{off} \tag{13}$$

4 Clamping inductive energy

Driving inductive loads without a freewheeling diode is possible. Care must be taken to ensure to not exceed the inductive energy capability of the driver at turn off. Excessive power dissipation due to high clamping energy can cause inelastic thermo-mechanical stress on the surface of the silicon. That is, the silicon expands rapidly and unevenly, heating up faster where the power dissipation is the highest. When the device cools, the silicon does not completely return to the original size. This phenomenon can be defined using the Coffin-Mason thermal fatigue model. This model defines the highest thermal gradient across the die without causing inelastic expansion to be approximately 60 °C.

Equation 14 – Coffin-Mason thermal fatigue model

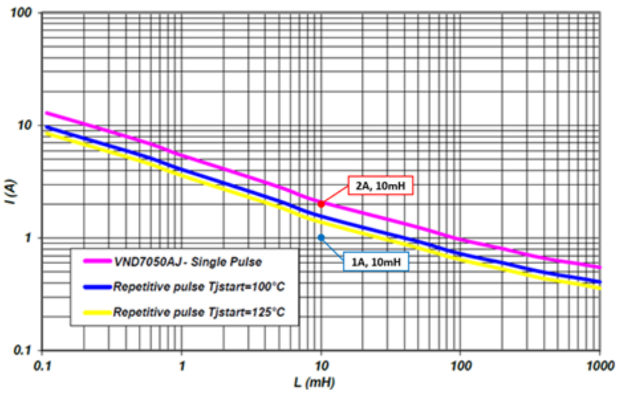
$$N_f = A_f{}^{-\alpha} \times \Delta T^{-\beta} \times G\left(T_{MAX}\right) \tag{14}$$

where:

- N_f is the number of cycles to failure
- f is the cycling frequency
- ΔT is the range of temperature during the cycle
- $G(T_{MAX})$ is an Arrhenius term evaluated at the max temperature reached during the cycle
- α is 2 (typically)
- β is 1/3 (typically)

As a result, this thermal gradient must be avoided. The best way to do this is to calculate the inductance and current and plot that on the current vs inductance plots given in the datasheets (see Figure 8).

Figure 8. **VND7050AJ maximum turn off energy plot**

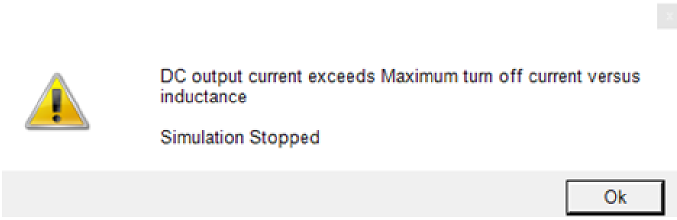


Looking at the Figure 8, anything below the lines are safe. For instance:

- A 1 A, 10 mH load is safe for both repetitive as well as single pulses. That is, this amount of energy does not degrade the device. However, repetitive pulses may add enough heat to overheat the part.
- A 2 A, 10 mH load exceeds the repetitive pulse limits. It is marginal for single events and should be verified in TwisterSIM.

If either of the parameters (current or inductance) are not inside the plot definitions, then you can run the simulation in TwisterSIM. Calculating the effects of clamping inductive energy is a short-term simulation and can be easily performed in TwisterSIM. If TwisterSIM determines that the load inductance is more than the device will be able to reliably handle, then a warning is displayed (see Figure 9).

Figure 9. **TwisterSIM excessive inductance warning**



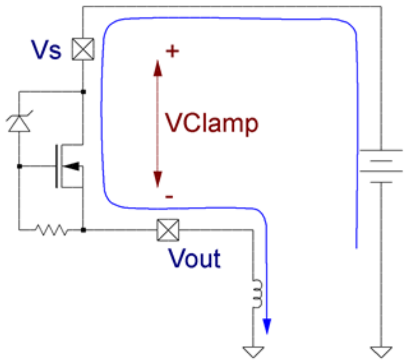
The TwisterSIM simulation results indicate if the junction temperature transition during the clamping function exceeds a safe level. If it does, then a freewheeling diode may be needed.

If repetitive clamping an inductive load is not too much inductive energy (see Figure 8), then the contribution to the thermal load can be estimated using a DC average. Then, the contribution to the thermal load can be estimated using a DC average. This is because the thermal peak is diffused into the driver long before the driver is turned back on.

DC average power (W) is done by calculating the energy (J) consumed in the driver and multiplying it by the frequency (J/s = W).

Calculating the energy during a clamp requires a bit more math than just calculating the energy in the inductor. This is because the driving circuit is also providing the clamping through the supply and not directly through ground back into the inductor (see Figure 10).

Figure 10. **Current flow during inductive clamp**



An s-domain equation can be written to describe the system in Figure 10. That equation can be transformed into the time domain by a Laplace transform. The resulting time domain equation is then solved for the current shown in Equation 15 below:

Equation 15 – Clamping current equation

$$I_{Clamp}\left(t\right) = \left(V_{Clamp} - V_{Batt}\right) - \left(V_{Clamp} - V_{Batt} - I_{OUT}R_I\right)e^{-\frac{R_I t}{L_I}} \tag{15}$$

Solving this for t_{Clamp} at 0 A provides the clamp duration:

Equation 16 – Clamping duration

$$I_{Clamp} = \frac{L_I}{R_I}\ln\left(1 - \frac{I_{OUT}R_I}{V_{Clamp} - V_{Batt}}\right) \tag{16}$$

Integrating the power (V_{Clamp} x I_{Clamp}(t)) over the clamping duration (t_{Clamp}) we obtain:

Equation 17 – Clamping energy

$$E_{Clamp} = \frac{L_I V_{Clamp}}{R_I{}^2}\left[\left(V_{Clamp} - V_{Batt}\right)\ln\left(\frac{V_{Clamp} - V_{Batt}}{\left(V_{Clamp} - V_{Batt} + I_{OUT}R_I\right)}\right) + I_{OUT}R_I\right] \tag{17}$$

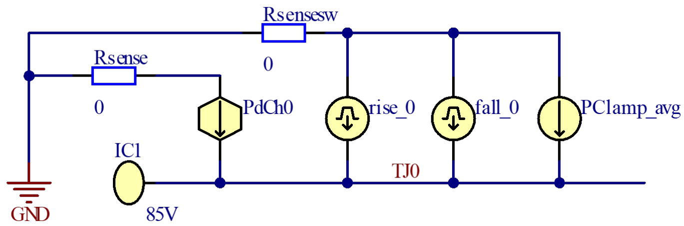
Multiplying this by the frequency obtains the average power dissipated by the clamping function.

Equation 18 – Average power dissipation due to repetitive clamping

$$P_{Clamp\_avg} = \frac{L_I V_{Clamp}}{R_I^2}\left[\left(V_{Clamp} - V_{Batt}\right)\ln\left(\frac{V_{Clamp} - V_{Batt}}{\left(V_{Clamp} - V_{Batt} + I_{OUT}R_I\right)}\right) + I_{OUT}R_I\right] freq \quad (18)$$

This can be added as a DC current source into the circuit:

Figure 11. Inserting the average power dissipation due to clamping



5 Supply current power dissipation calculations (Psupply)

Losses due to quiescent current are simply the supply voltage multiplied by the supply current. Care must be taken to choose the correct current from the datasheet. The current to use is $I_{GND(ON)}$. This parameter defines the current coming out of the ground pin while the outputs are under a load. The supply current changes with load current. This is because there is current out of the multisense pin that mirrors a portion of current from the high side switch. The portion of current from the high side switch shows up on the ground pin. As a result, the ground current changes depending on the output current(s).

Table 4. Supply current parameter, $I_{GND(ON)}$ (VND7050AJ)

Symbol	Parameter	Test conditions	Min.	Typ.	Max.	Unit
$I_{GND(ON)}$	Control stage current consumption in ON state. All channels active.	V_{CC} = 13 V, V_{SEn} = 5 V, V_{FR} = $V_{SEL0, 1}$ = 0 V, V_{IN0} = 5 V, V_{IN1} = 5 V, I_{OUT0} = 2 A, I_{OUT0} = 2 A	-	-	12	mA

This power dissipation is created in the control section of the die. As a result, the Psupply current is injected after the channel-only elements in the Foster model. That is, the current (power) is inserted before the R3, C3 element pair (refer to Figure 2).

6 Programming the elements (summary)

This section provides a summary of the different elements used in Figure 2.

6.1 Conduction losses definitions

Current definition has two components. First to describe the load current accurately then to calculate the power as a result of that current.

6.1.1 Current definition for conduction losses

To accommodate for the rise time the current is delayed by the rise time. This allows the switching losses at turn-on not to overlap with the conduction losses.

PWM current parameters (for our example ILOAD_Ch0):

- Initial value is 0
- Pulsed value is application current (the example in Figure 2 uses 5 A)
- Time delay is t_{rise} (from Equation 8 above)
- Rise time is 0 s
- Fall time is 0 s
- Pulse width is duty/PWM frequency
- Period is 1/PWM frequency

A SPICE PWM load current definition (example) is:

```
ILOAD_Ch0 0 SNS0 DC 0 PULSE(0 5 45us 0s 0s 5.92ms 8.33ms) AC 1 0
```

Piecewise linear current definitions are typically used for on/off loads where the current changes naturally during the course of driving it. A good example would be a motor load current definition. This definition would have inrush, run, and stall currents while the switch is only turned on at the beginning and off at the end.

A SPICE PWL load current definition (example) is:

```
ILOAD_Ch1 0 SNS1 DC 0 PWL(0 0 3 0 3.01 5 10 5) AC 1 0
```

6.1.2 Power calculations for conduction losses

The power calculation is a simple equation illustrated in Section 2.2.2 . The only changes to this equation are done to accommodate for the different $R_{DS(on)}$ values in the different switches. The below example shows a different nodal connection for the PWMed output as sense resistors (Rsense and Rsensesw) were added to enable visibility to the power injected into the junction.

The power dissipation calculation SPICE command using pre-defined load currents (example) is:

```
BPdCh0 NetPdCh0_1 TJ0 I=V(SNS0)^2*0.050*(0.8+0.008*V(TJ0))
BPdCh1 0 TJ1 I=V(SNS1)^2*0.05*(0.8+0.008*V(TJ1))
```

6.2 Switching losses definitions

6.2.1 Rise time definitions

The rise time starts off the simulation. There is no delay associated with it.

t_{rise} parameters (for our example rise_0) are:

- Initial value is 0 s
- Pulsed value is P_{Sw_on} as calculated in Equation 10
- Time delay is 0 s
- Rise time is 0 s
- Fall time is 0 s
- Pulse width is t_{rise} (from Equation 8)
- Period is 1/frequency

The rise time power SPICE command line (example) is:

```
Irise_0 Netfall_0_1 TJ0 DC 0 PULSE(0 7.3333 0 0s 0s 45us 8.33ms) AC 1 0
```

6.2.2 Fall time definitions

The fall time is delayed by both the rise time and the duty cycle (on time).

t_{fall} parameters (for proposed example fall_0) are:

- Initial value is 0
- Pulsed value is P_{Sw_off} as calculated in Equation 11
- Time delay is duty/frequency + t_{rise}
- Rise time is 0 s
- Fall time is 0 s
- Pulse width is t_{fall} (from Equation 9)
- Period is 1/frequency

The fall time power SPICE command line (example) is:

```
Ifall_0 Netfall_0_1 TJ0 DC 0 PULSE(0 7.2941 5.965ms 0s 0s 42.5us 8.33ms) AC 1 0
```

6.3 Supply current power dissipation definition

This is a simple current source. The value 0.162 is a result of 13.5 V x 12 mA.

The supply current power dissipation SPICE command line (example) is:

```
IPsupply 0 NetC2_2 0.162
```

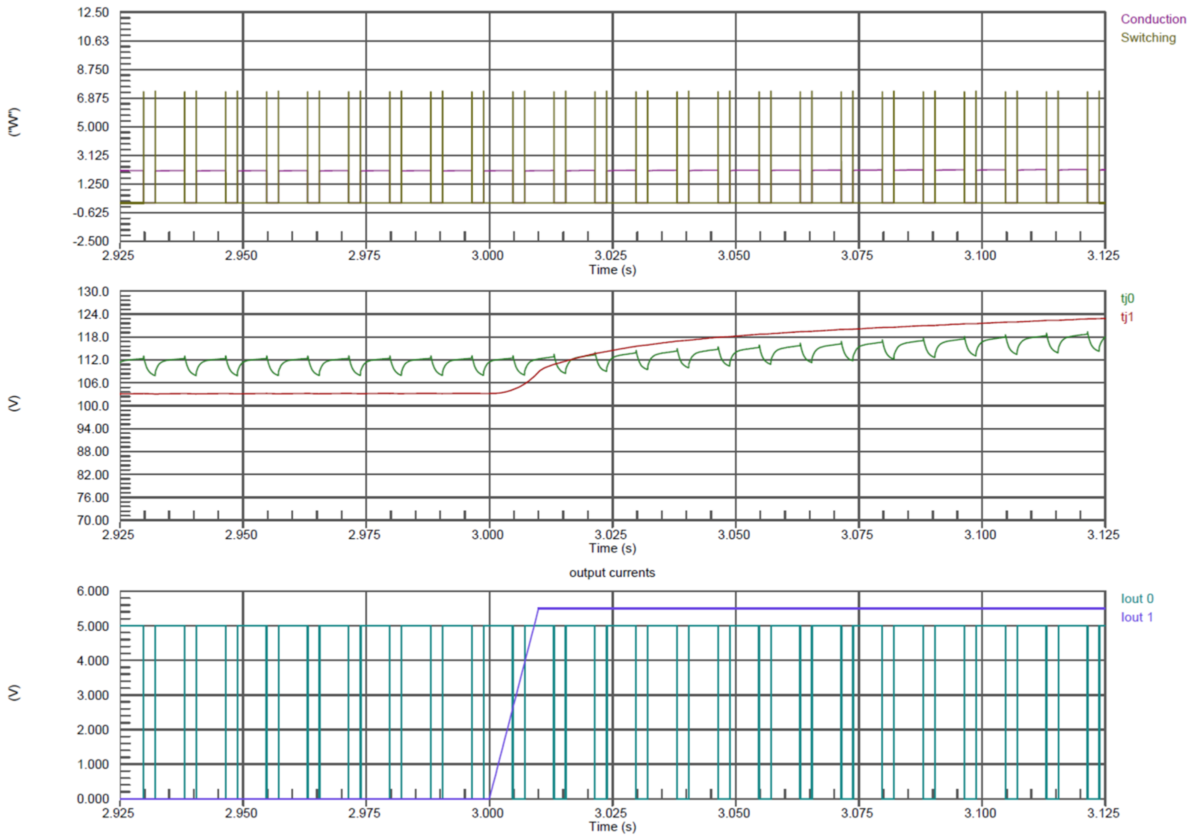
7 Reading the results

The results from the SPICE simulation provide more than the junction temperature. At the same time the results are limited in being able to reflect any of the thermal protection functionality (see AN5368 for details on VIPower protection strategies).

The results reflect the junction temperature at a given moment for sure. However, this simulation does not reflect thermal interventions such as power limitation or thermal shutdown. Nor does it reflect current limit.

The Figure 12 illustrates a portion of a 10 s simulation. This section illustrates the output 1 transition from off to on. In this plot two junction temperatures are inserted onto one plot, the two output currents onto one plot and the two power dissipation components of OUT0 onto one plot. The power dissipation for OUT1 is not shown.

Figure 12. Simulation results



From Figure 12 it can be seen that there are no dramatic thermal excursions. If there were, it would be best to measure the temperature difference between the junction and the node between R2 and R3 where the quiescent current power is injected (see Figure 2). From this delta it can be estimated if the device enter power limitation. If the difference between these two nodes is anything close to 60 °C then it would be advisable to rerun this portion of the simulation in TwisterSIM. TwisterSIM has a more complex thermal model than what is in the datasheet. The TwisterSIM model accommodates for the power limitation function more accurately. Using the difference between the junction (TJx) and node between R2 and R3 as an indicator of power limitation is only an approximation.

If the junction temperature ever exceeds 150 ºC then we can assume that the part can enter thermal shutdown.

The advantage of using SPICE to simulate the thermal response is that it can take less time for longer duration simulations without affecting accuracy. It can also allow for more complex configurations. It may take less time to run your simulation in TwisterSIM that it would for you to walk through this lengthy application note and implement this into your spice simulator. However once done it is available for the next design.

Appendix A Code for the simulations shown

The SPICE netlist for Figure 2 generated by Altium (comments in red added by the author) is:

```
Junction Temperature VND7050AJ 4L with Ron adjust
*SPICE Netlist generated by Advanced Sim server on 1/10/2020 11:55:34 AM

*Schematic Netlist:
* FOSTER Capacitances
C1 TJ0 NetC1_2 0.00035
C2 NetC1_2 NetC2_2 0.005
C3 NetC2_2 NetC3_2 0.05
C4 NetC3_2 NetC4_2 0.4
C5 NetC4_2 NetC5_2 4
C6 NetC5_2 NetC6_2 18
C7 TJ1 NetC7_2 0.00035
C8 NetC7_2 NetC2_2 0.005

*tfall pulse
Ifall_0 Netfall_0_1 TJ0 DC 0 PULSE(0 7.2941 5.965ms 0s 0s 42.5us + 8.33ms) AC 1 0

*initial conditions
.IC V(TJ0)=85V
.IC V(TJ1)=85V

*Load current definitions
ILOAD_Ch0 0 SNS0 DC 0 PULSE(0 5 45us 0s 0s 5.92ms 8.33ms) AC 1 0
ILOAD_Ch1 0 SNS1 DC 0 PWL(0 0 3 0 3.01 6 10 6) AC 1 0
BPdCh0 NetPdCh0_1 TJ0 I=V(SNS0)^2*0.050*(0.8+0.008*V(TJ0))
BPdCh1 0 TJ1 I=V(SNS1)^2*0.05*(0.8+0.008*V(TJ1))

*losses due to the power supply
IPsupply 0 NetC2_2 0.192

*FOSTER resistances
R1 TJ0 NetC1_2 1.8
R2 NetC1_2 NetC2_2 3.2
R3 NetC2_2 NetC3_2 6
R4 NetC3_2 NetC4_2 4
R5 NetC4_2 NetC5_2 3
R6 NetC5_2 NetC6_2 7
R7 TJ1 NetC7_2 1.8
R8 NetC7_2 NetC2_2 3.2
*trise pulse
Irise_0 Netfall_0_1 TJ0 DC 0 PULSE(0 7.3333 0 0s 0s 45us 8.33ms) AC 1 0

*sense resistors for the conduction losses calculators
Rs0 0 SNS0 1
Rs1 0 SNS1 1

*Sense resistors added to measure “power” into the device.
Rsense 0 NetPdCh0_1 0
Rsensesw 0 Netfall_0_1 0

*Ambient temperature setting
VTAmb NetC6_2 0 85V
```

```
.SAVE 0 NetC1_2 NetC2_2 NetC3_2 NetC4_2 NetC5_2 NetC6_2 NetC7_2 Netfall_0_1
.SAVE NetPdCh0_1 SNS0 SNS1 TJ0 TJ1 Ifall_0[v] ILOAD_Ch0[v] ILOAD_Ch1[v] IPsupply[v]
.SAVE Irise_0[v] VTAmb#branch @VTAmb[z] @C1[i] @C2[i] @C3[i] @C4[i] @C5[i] @C6[i]
.SAVE @C7[i] @C8[i] @R1[i] @R2[i] @R3[i] @R4[i] @R5[i] @R6[i] @R7[i] @R8[i] @Rs0[i]
.SAVE @Rs1[i] @Rsense[i] @Rsensesw[i] @C1[p] @C2[p] @C3[p] @C4[p] @C5[p] @C6[p] @C7[p]
.SAVE @C8[p] @Ifall_0[p] @ILOAD_Ch0[p] @ILOAD_Ch1[p] @IPsupply[p] @Irise_0[p] @R1[p]
.SAVE @R2[p] @R3[p] @R4[p] @R5[p] @R6[p] @R7[p] @R8[p] @Rs0[p] @Rs1[p] @Rsense[p]
.SAVE @Rsensesw[p] @VTAmb[p]

*PLOT TRAN -1 1 A=NetC1_2
*PLOT OP -1 1 A=NetC1_2

*Selected Circuit Analyses: duration and step size of simulation
.TRAN 0.0001666 10 0 0.0001666
.OP

.END
```

Appendix B Alternative $R_{DS(on)}$ equation

There is an alternative $R_{DS(on)}$ equation to the simple linear equation (referencing Equation 1).

The Equation 1 is a linear equation that is fairly accurate for +6σ distribution data and remains within the specification parameters. This equation is best to use when calculating worst case hot temperature (above 25 °C) analysis. However, a more accurate equation is available for the typical $R_{DS(on)}$ overtemperature. This last equation provides the typical $R_{DS(on)}$ curve that works over the full temperature range from -40 °C to 150 °C.

Equation 19 – Typical $R_{DS(on)}$ overtemperature

$$R_{DS(on)}\left(T_J\right) = R_{DS(on)}\ @\ 25\ ºC\left[1 + \frac{T_J + 125\ ºC}{300\ ºC}\right]^{1.6} \tag{19}$$

For typical $R_{DS(on)}$ over the temperature range, this equation is the best fit.

Appendix C Acronyms, abbreviations and reference documents

Table 5. Acronyms and abbreviations

Terms	Description
Conduction losses	Power dissipation in a switch while it is on and conducting current into the load
EMI	Electromagnetic interference
Foster model	A thermal modeling method using resistors and capacitors to emulate the dynamic thermal response of a semiconductor
IC	Integrated circuit
IPWL	A SPICE function that generates a piecewise linear current source
M07	Seventh generation VIPower technology
PWL	Piecewise Linear, a line function that describes a line by defining successive points and connecting them with straight lines
PWM	Pulse width modulation
PWMmed	Pulse width modulated
$R_{DS(on)}$	On resistance of an enhanced MOSFET
Slew rate	The rate of change in voltages or current when switching from one state to another (on to off or off to on)
SPICE	Open-source software that simulates the operating conditions of analog circuits. It is short for 'Simulation Program with Integrated Circuit Emphasis'.
Switching losses	Power dissipation in a switch while it is transitioning between off to on or on to off
TwisterSIM	Thermal tool made by ST for thermal simulations of ST VIPower products
VIPower	Vertical Intelligent Power, a semiconductor process developed by ST that is composed of a power MOSFET with embedded control circuitry
W_{on}, W_{off}	Switching energy parameters used to calculate switching losses in ST high side drivers. They represent the amount of energy dissipated in a single switching event (on or off)

Table 6. Reference documents

Document name	Document type
UM1874	TwisterSIM user manual
AN5368	VIPower M0-7 thermal protections

Revision history

Table 7. **Document revision history**

Date	Version	Changes
19-Apr-2021	1	Initial release.

Contents

List of tables

List of figures