

Introducing NI Multisim 12.0 Power Components

Overview

NI Multisim is a powerful tool used to simulate and prototype power electronics circuit designs. NI has added new power components models to Multisim 12.0 that, along with the existing power SPICE models from various semiconductor manufacturers, help you evaluate the different power supply circuits of different scales at an early design stage. You can use models for AC/DC converters, switch-mode power supplies, and machine, sensor, motion control, and other electromechanical devices to evaluate power electronics systems ranging from the simple power supplies, converter circuits, and motor controllers used in automotive applications to power grid systems at an early design stage.

This tutorial provides an overview of the new power components in the Multisim database and the types of designs you can use them in. It also includes a brief description of possible analysis problems that can occur during the SPICE simulation of power circuits.

The tutorial reviews two example designs of an isolated 2 W bias supply circuit and a 12 V forward converter developed by ON semiconductor based on the company's power supply controller parts and simulated in the Multisim design environment. This paper emphasizes how the new power components optimize your design and improve productivity by helping you avoid time-consuming, unsuccessful design iterations that take up to five days each to cover simulation, prototyping, and testing and validation.

Table of Contents

1. [New Power Components in Multisim 12.0](#)
2. [Application Areas](#)
3. [Application Examples](#)
4. [Recommendations for Power Supply Simulation](#)
5. [Additional Resources](#)

New Power Components in Multisim 12.0

NI has updated the Multisim components database with 92 new component models that are widely used in the design of power and electromechanical devices. This database features more than 1,000 power component models; metal-oxide semiconductor, field-effect transistors (MOSFETs); switching controllers; diodes; IGBTs; SCR transformers; and other components that, along with the 22,000 components in the Multisim database (including models for basic and advanced analog parts), can help you optimize your power and electromechanical design at the SPICE simulation stage. The new models include the following:

- Power switches: New generic models for diodes, gate turnoffs, silicon-controlled rectifiers, TRIACs, transistors, and transistors with a body diode were added to the Multisim database. You can change the parameters of these generic models to model the performance of power electronics devices that do not have SPICE models at the early stage of schematic capture and simulation.
- Power controllers: New generic models of phase angle controllers and PWM controllers were added to offer the possibility of modeling and simulating these vital devices in the design of high-efficiency high-speed power converter applications.
- Reconfigurable transformers: These generic models of configurable transformers were added to the database to offer the possibility of modeling the behavior of almost any transformer in the market.
- Nonideal RLCs: These help improve the modeling of resistive losses and reactive parasitic effects of circuit components.

You can find these components in the master database under the power and the basic groups options.

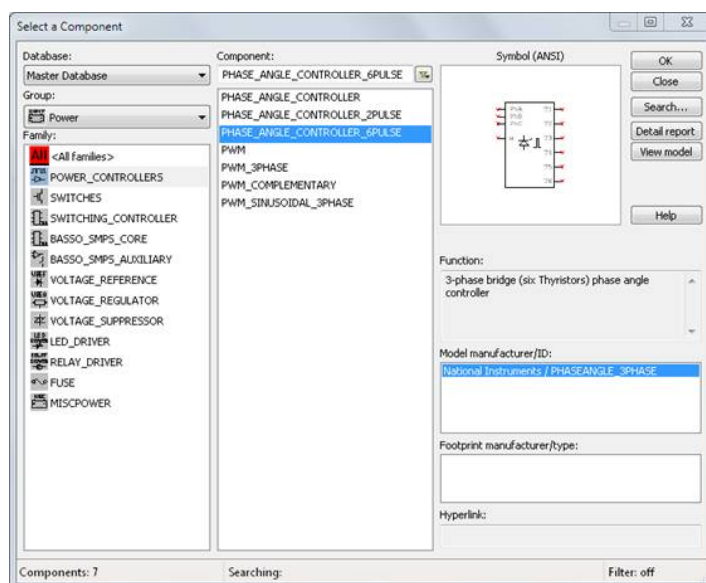


Figure 1. New Power Components in Multisim 12.0 Database

Application Areas

In newer power device designs, specifications become more and more stringent every day. Higher efficiency, higher speed, and accurate control require powerful simulation capabilities and an optimized design environment to help designers become more productive in their prototyping approaches. The capability of modeling these power components in Multisim takes advantage of the design and simulation fidelity of circuits like single-phase and 3-phase rectifiers; inverter circuits; and buck, flyback, and forward converters.

These circuits are extremely important in every power electronics application from the smaller power supplies used in the automotive industry or PCs to the large-scale converters that are critical in connecting wind turbines and solar cells to power grids or in using high-power telecom transmitters.

Below are snippets of some circuits considered to be the building blocks of any power conversion design. A snippet is a PNG image that you can simply drag and drop into Multisim to load the design.

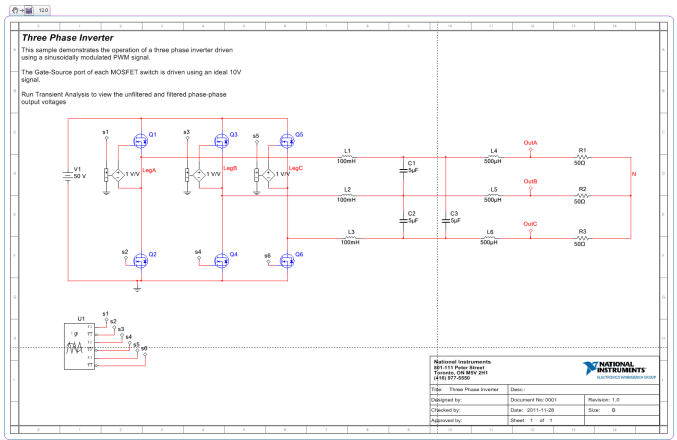


Figure 2. 3-Phase Inverter Using the New Models for Pulse-Width Modulation (PWM) Controllers, Nonideal RLCs, and Power Transistors

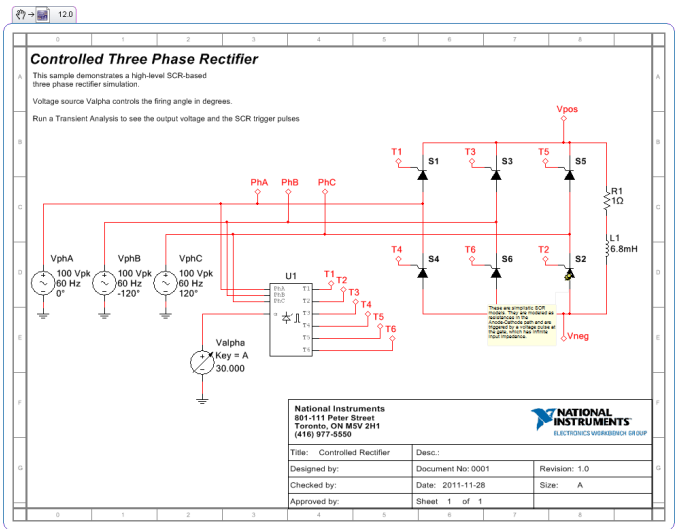


Figure 3. 3-Phase Rectifier Using the New Models for SCRs, Phase Angle Controllers, and Nonideal Resistors

Application Examples

This section of the tutorial examines two applications that benefit from the new power component models in Multisim 12.0.

2 W Isolated Bias Supply

In this application, the circuit is a 2 W bias supply that steps down an input ranging from 36 V to 78 V to an output of 12 V. This application is widely used in automotive circuits as well as telecommunications systems.

The circuit design is based on the ON Semiconductor model of integrated circuit NCP1030, which is a highly integrated power switch circuit. The part symbol, model, and footprint are available in Multisim, and the theoretical background and design steps are explained in detail in ON Semiconductor application note [AND8119/D](#).

The Figure 4 snippet (a PNG image that you can drag and drop into Multisim to directly load the design) shows the layout that includes new components of nonideal RLC models, rectifiers, and configurable transformers.

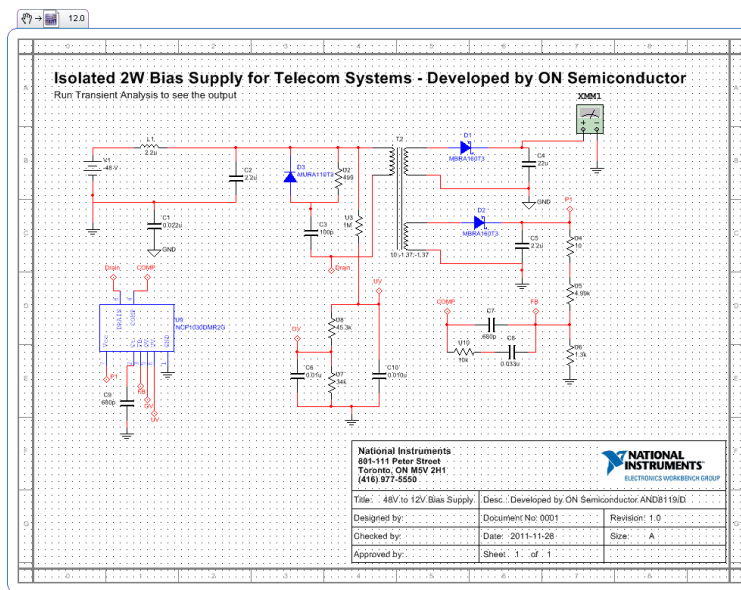


Figure 4. 48 V to 12 V Bias Supply in Multisim

The transient analysis of the design results is shown in Figure 5.

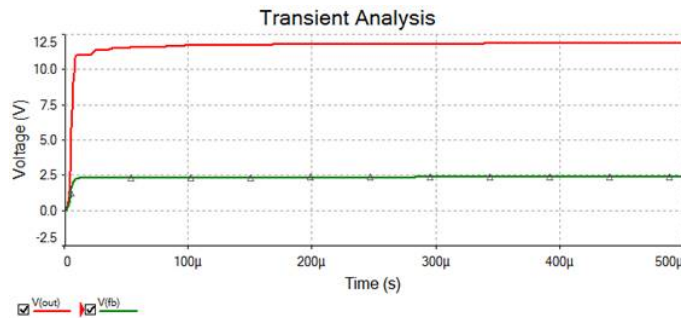


Figure 5. Transient Analysis Results of the 48 V to 12 V Bias Supply in Multisim

The transient helps you analyze the circuit by showing a buildup of time around 10 μs for the regulated voltage to stabilize at 12 V. The RLC components help you model the nonidealities and the parasitic effects associated with all impedances. The switching controller is based on the SPICE model provided by ON Semiconductor. Once you have accurately evaluated the design of this circuit using Multisim, the circuit is ready for prototyping with minimized errors in the desired specifications.

A Simple 12 V, 22 W Offline Forward Converter

This forward converter developed by ON Semiconductor is based on controller NCP1028, which is modeled in Multisim. NCP1028 is a high-voltage switching regulator for medium power offline switched-mode power supplies (SMPSs). This circuit provides 12 V output at up to 2 A peak output current over the entire universal input range (90 to 265 VAC), which makes it suitable for almost any application that involves power supply design.

Learn more about the technical design details in ON Semiconductor application note [AND8489/D](#).

The following models of power components in Multisim 12.0 were used in the design:

1. Configurable transformers
2. Diode rectifiers
3. Nonideal RLCs
4. Optocouplers
5. NCP1028 controllers

Figure 6 is a snippet of the design.

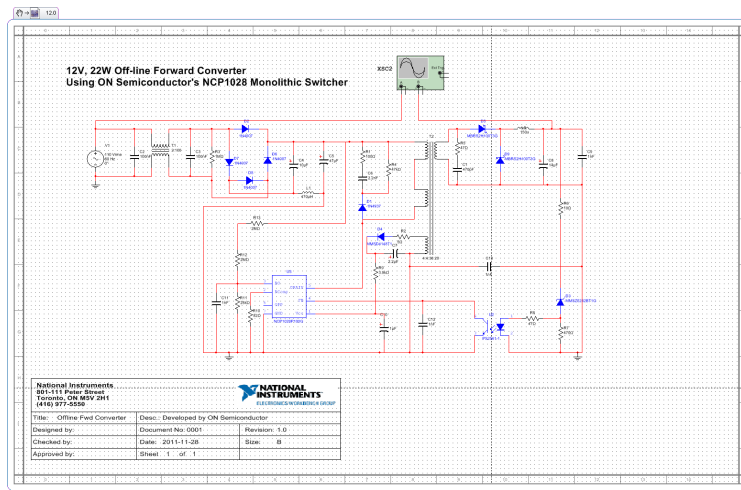


Figure 6. 12 V AC to DC Forward Converter Schematic

You also can show the transient response of this circuit on the oscilloscope instrument built into Multisim (see Figure 7).

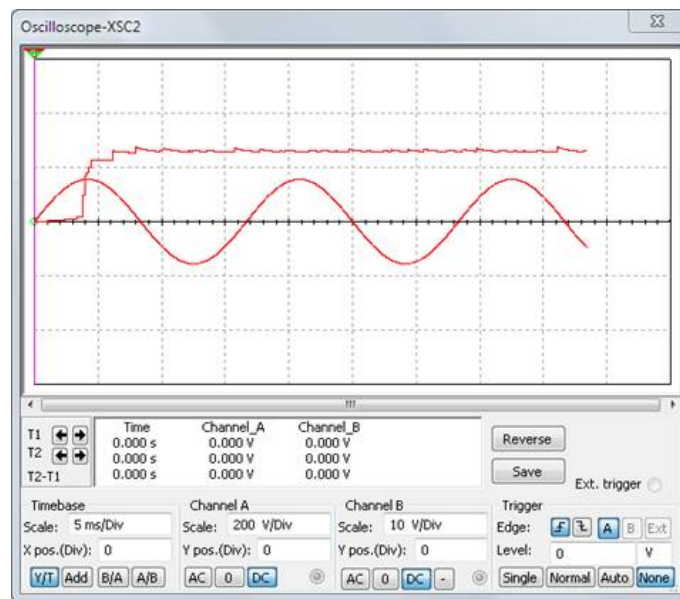


Figure 7. Measured Transient Response on the Built-In Virtual Oscilloscope Instrument of Multisim

As the oscilloscope measurement shows, an AC input voltage of 180 V peak amplitude is regulated to a DC level of 12 V. The setting time of this regulator circuit is less than 10 ms. You can perform further advanced analyses to evaluate the ripple level of the DC output, the efficiency of the circuit, and the sensitivity of the circuit to parametric variations. The nonideal RLC and the configurable transformers help to accurately model all of the parasitic effects while the SPICE model of the switching control provided by ON Semiconductor almost guarantees the alignment of this simulation with a fabricated physical prototype.

Recommendations for Power Supply Simulation

The most informative analysis for power circuit simulation is transient analysis. You can use it to visualize the transition of voltages at the nodes from one state to another and illustrate how the circuit converges to its steady state. You need to understand the voltage potential built up on energy storing elements such as capacitors and inductors (elements charging and discharging) for design specifications and performance.

Setting an accurate transient simulation requires basic knowledge of SPICE simulation fundamentals. Transient simulation starts with a DC operating point solution with all of the time-varying elements set to an initial voltage and current values and replaced by their corresponding linear models.

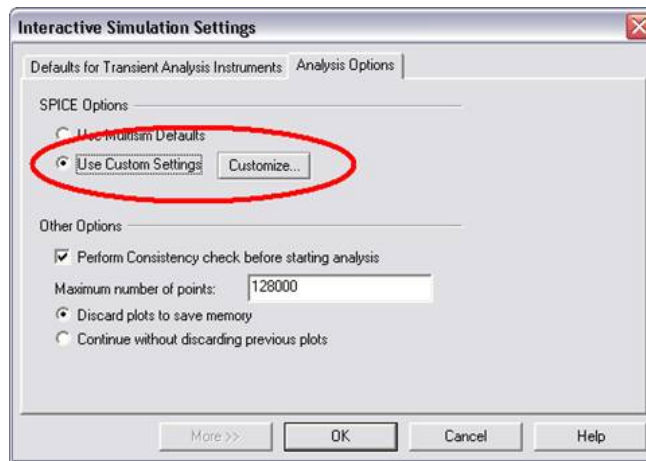


Figure 8. Interactive Simulation Settings

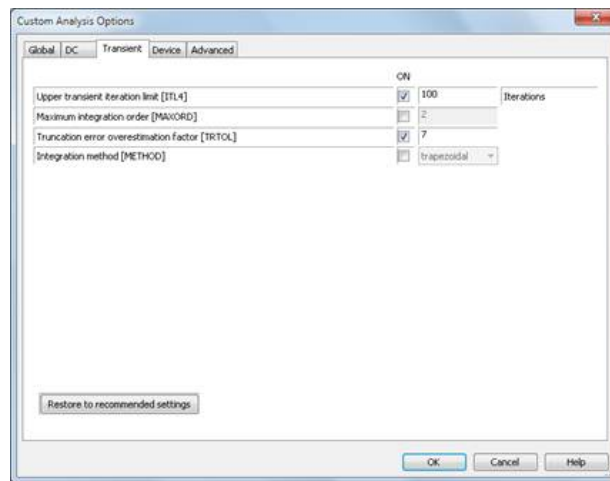


Figure 9. Modifiable SPICE Simulation Parameters of the Transient Analysis

You need to set the Table 1 simulation parameters appropriately if the circuit fails to converge. You can find these under **Simulate/Interactive Simulation Settings** on the **Analysis options** tab. Use custom settings and click the **Customize** box.

Parameter	Function	Action
RELTOL	Relative Error Tolerance	<ol style="list-style-type: none"> 1. Increase for a more accurate simulation 2. Decrease for a faster simulation
GMIN	Minimum circuit conductance	<ol style="list-style-type: none"> 1. Increase to help a converging solution
IC	Initial node voltages	<ol style="list-style-type: none"> 1. Set all charging capacitors voltages to zero
ITL	Upper Transient Iteration Limit	<ol style="list-style-type: none"> 1. Increase to help a circuit converge
METHOD	Time Step Integration Method	<ol style="list-style-type: none"> 1. Change and make sure results remain consistent

Table 1. Simulation Parameters That Must Be Set If the Circuit Fails to Converge

You must set the simulation time, the time step, and the initial conditions to values before running the simulation, as shown in Figure 10.

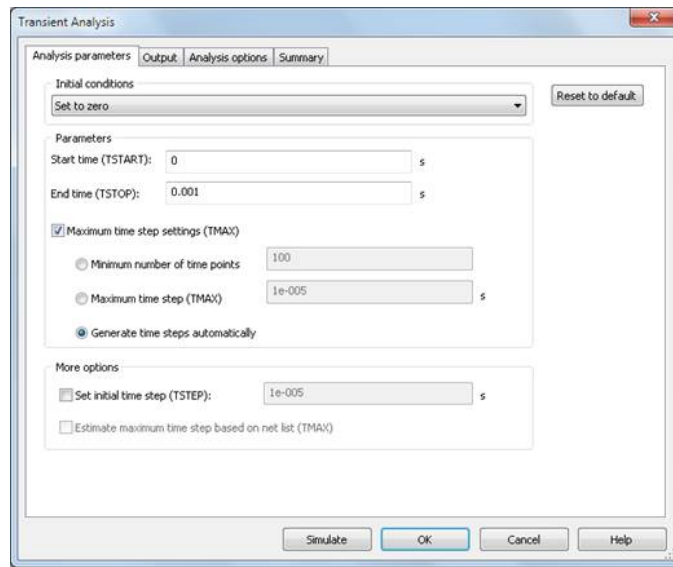


Figure 10. Transient Analysis Setup

Once the transient analysis is set appropriately and the circuit is functioning to specifications, you can perform different analyses, such as Fourier, DC sweep, and sensitivity, to evaluate other responses and confirm the design applicability.

Read more about different types of [analyses in Multisim](#).

Learn about the [fundamentals of SPICE simulation](#).

Additional Resources

[Reference Design: System Simulation of a 3-Phase Inverter Energy Application in Multisim and LabVIEW](#)

[Reference Design: Implementation of a DC Motor H-Bridge Circuit and Control Logic Using Multisim and LabVIEW FPGA](#)

[Solutions for Green Engineering from NI](#)

Legal

This tutorial (this "tutorial") was developed by National Instruments ("NI"). Although technical support of this tutorial may be made available by National Instruments, the content in this tutorial may not be completely tested and verified, and NI does not guarantee its quality in any way or that NI will continue to support this content with each new revision of related products and drivers. THIS TUTORIAL IS PROVIDED "AS IS" WITHOUT WARRANTY OF ANY KIND AND SUBJECT TO CERTAIN RESTRICTIONS AS MORE SPECIFICALLY SET FORTH IN NI.COM'S TERMS OF USE (<http://ni.com/legal/termsofuse/unitedstates/us/>).