AN214
MPSmart User Guide

by Ida Li

September 2023

TABLE OF CONTENTS

INSTRUCTIONS

This user guide is to help users understand and utilize the powerful features and capabilities of MPSmart software. MPS distributes the MPSmart simulation tool under license from SIMPLIS Technologies. This software integrates SIMetrix and SIMPLIS simulators, providing users with a comprehensive simulation and analysis tool for MPS products.

Overview of MPSmart

MPSmart is a powerful software tool that allows users to select and download suitable MPS IC models, simulate the chosen IC models, and analyze the performance of their designs. With its user-friendly interface, MPSmart makes it easy for users to develop advanced designs with using large library of MPS recommended schematics and models.

MPSmart includes two simulators, SIMetrix and SIMPLIS. SIMetrix is a fast and accurate SPICE type simulator. SIMPLIS is a circuit simulator specifically designed to handle the simulation challenges of switching power systems. Most MPS product IC models are created with SIMPLIS. This application note will mainly introduce MPSmart models built on SIMPLIS simulator.

SIMPLIS supports three types of simulations, POP (Periodic Operating Point) simulation, frequency-based AC simulation, and transient simulation. All the simulations will be introduced in the Running a Simulation section on page 12.

Features and Benefits

MPSmart offers several key features and benefits for users, which are listed below:

- Allows users access to a wide range of MPS IC models, allowing users to design and optimize diverse power electronic controllers.
- Provides support for useful simulation types. Aside from normal time domain-based transient analysis, MPSmart provide small-signal analysis for the switching circuit, so there is no need for error-prone average modes.
- Gives users full schematic capture and editing capabilities.
- Provides advanced applications and customization options. The MPS Design Assistant spreadsheet and MPS GUI expand the MPSmart application range. See the Advanced Applications section on page 16 for more details.
- Includes comprehensive troubleshooting and support resources.

System Requirements

MPSmart operates on Microsoft Windows platforms. MPSmart is fully supported by Windows 10 (64-bit) and above.

Please note that the performance of MPSmart may be affected by computer hardware and complexity of designs. It is recommended to use a system with higher specifications for large or complex projects.

GETTING STARTED

This section describes the process of downloading, installing, and launching MPSmart for the first time, and allows the user to become familiar with the user interface

Downloading, Installing, Launching, and Upgrading MPSmart

Download and install the MPSMart software on the computer. The MPSmart software can be downloaded from the MPS website: https://www.monolithicpower.com/en/mpsmart-v8.html.

After downloading and installing MPSmart on the computer, launch MPSmart by double-clicking the shortcut on the desktop or in the Start menu. Figure 1 shows the MPSmart main window.

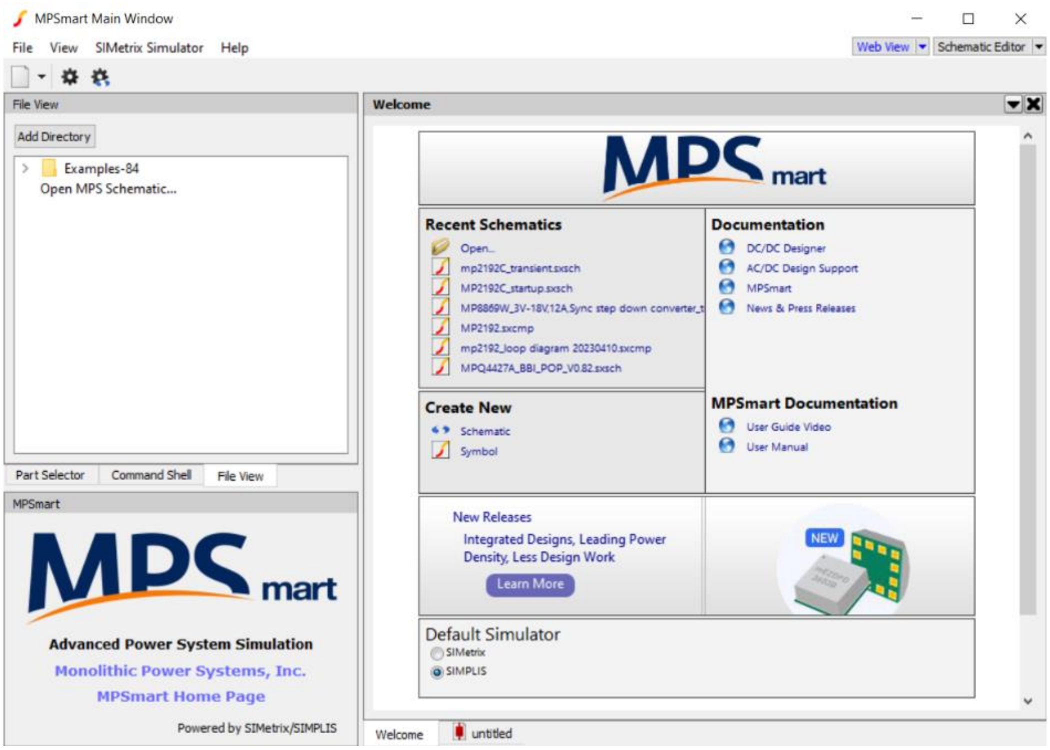


Figure 1: MPSmart Main Window

MPSmart is upgraded at least once a year with updated features and the latest MPS product model libraries. Users can check for updates or reinstall the software to receive full support.

User Interface Overview

The main user interface consists of two parts: System View and Workspace View. On the left side of the MPSmart window, System View provides tools to operate the program. Workspace View provides a means for developing and reviewing designs, including the schematic editor and waveform viewer.

Figure 2 shows an overview of the user interface.

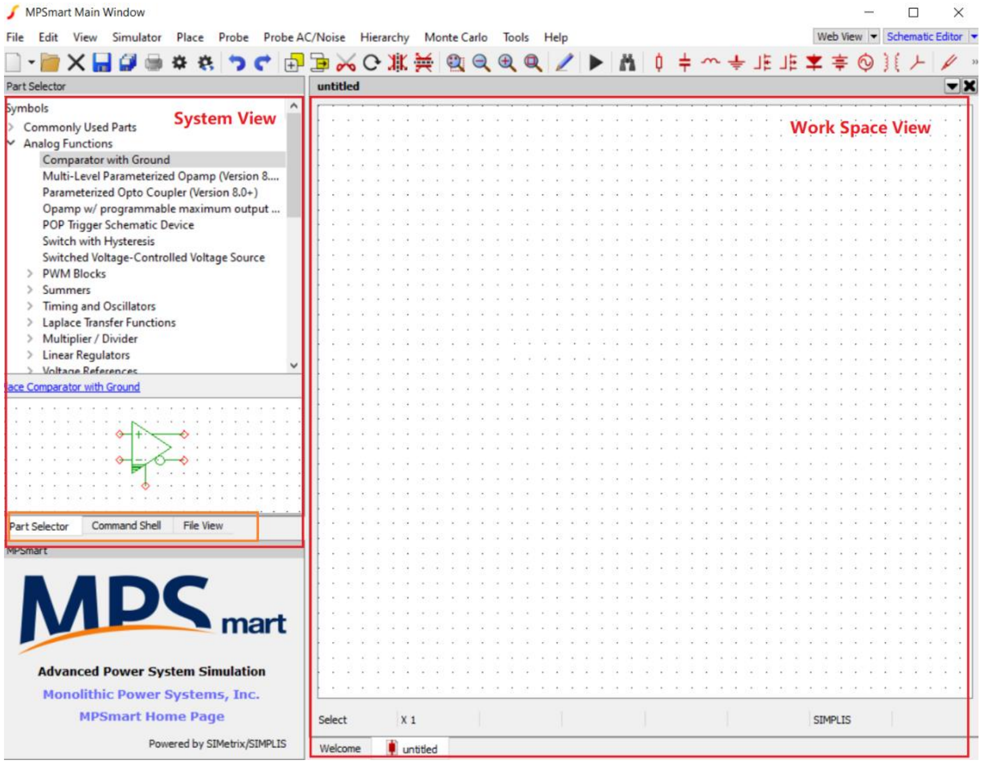


Figure 2: Overview of the User Interface

Figure 3 shows the sections of System View, which include Part Selector, Command Shell, and File View.

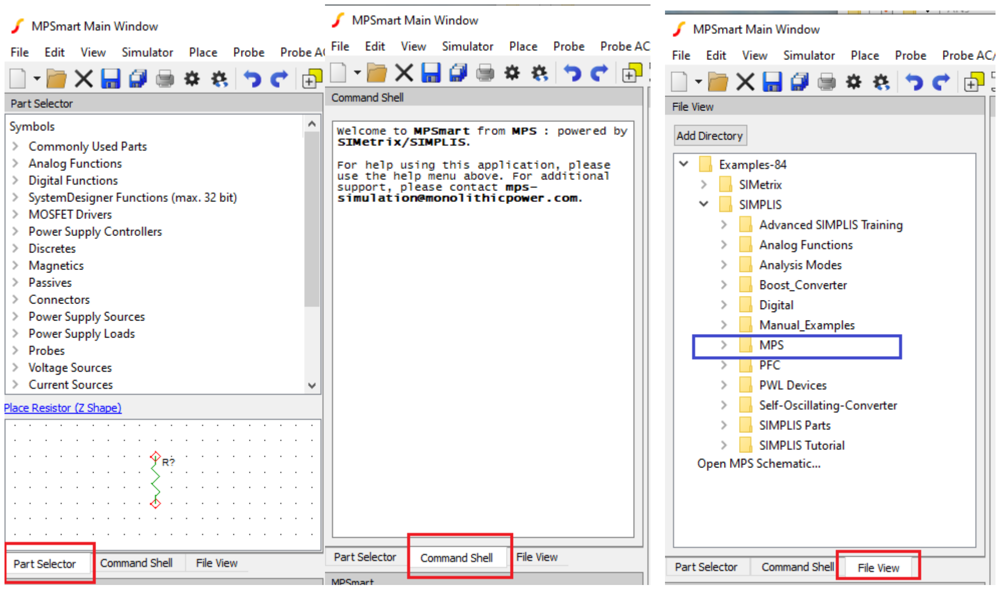


Figure 3: System View (Part Selector, Command Shell, and File View)

Part Selector: Part Selector covers most of the component categories. Users can select components from the Part Selector section to place on the schematic to start a design.

Command Shell: Command Shell displays the status of the software performing actions or executes scripts.

File View: File View displays a hierarchical list of files and folders. “Examples” is the default folder that is automatically installed with MPSmart. In the default example folder, click the folder titled “MPS” that contains the MPS product model.

Under File View, click the “Add Directory” button to choose the frequently accessed folder to add it to the File View list. Users can also open, rename, delete, or manage the folders and files in the File View panel.

Figure 4 shows Workspace View, which provides the means to develop and review designs, including the Schematic Editor and Waveform Viewer.

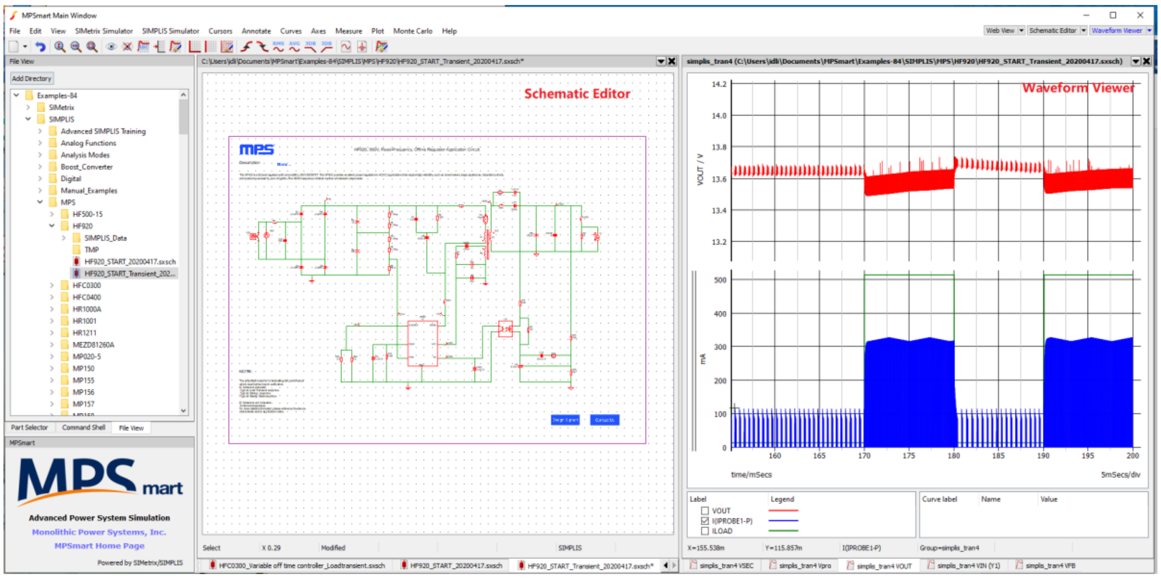


Figure 4: Schematic Editor and Waveform Viewer

Schematic Editor: Users can create and modify circuit designs in Schematic Editor. This editor allows users to place components, connect wires, and adjust component values. Users can also right-click on components within Schematic Editor to access additional options and commands, such as disabling components, editing and adding properties, or editing and updating symbols.

Waveform Viewer: Waveform Viewer is a dedicated area for viewing and analyzing the waveforms generated during simulations. Users can use Waveform Viewer to check the performance of the design, compare different simulation runs, and measure critical parameters (e.g. voltage, frequency, and duty cycle).

These user interfaces make it easy to navigate MPSmart and access its key features. The following sections of this manual provide detailed instructions on how to use MPSmart's various tools and functions.

SELECTING AND DOWNLOADING MPS IC MODELS

This section guides users through the process of selecting and downloading MPS IC models for use in designs. It discusses how to access the MPSmart example folders, browse the product page, download MPS IC models, and get help from MPS support teams.

Accessing MPSmart Example Folder

As mentioned in the User Interface Overview section on page 4, users can access MPS IC models through the MPSmart example folder, which is automatically installed with MPSmart. Follow the steps below to access the IC models in MPSmart:

1. Launch MPSmart and open File View (located on the left side of the window).
2. The folder titled "MPS" contains pre-installed IC models, typical schematics, and example projects for various MPS products (see Figure 5). Expand and browse the file's contents to find the desired MPS IC model example project.
3. Double-click on the desired IC model schematic to open it in Schematic Editor.

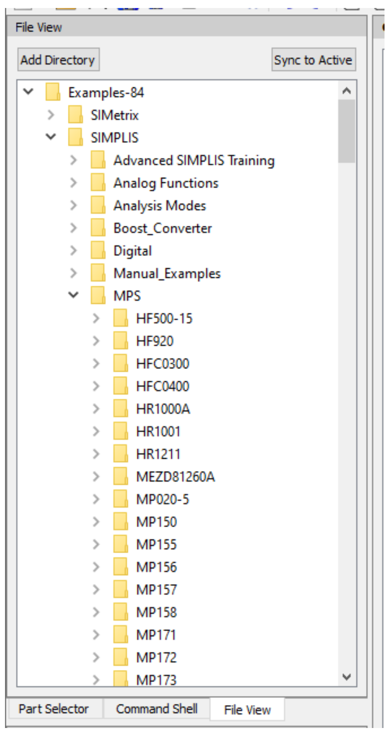


Figure 5: MPSmart Example Folder

Browsing on the MPS Website

To find the appropriate MPS products for the application and to download the correct IC model, visit the MPS website: https://www.monolithicpower.com. Use the search bar to enter the part number of the desired MPS IC, or navigate to the Products tab in the main menu to view the product categories and access the corresponding product pages.

Figure 6 on page 8 shows an example of how to find MPSmart models on the MPS website. For example, to find the MPM3595-25's MPSmart model, enter the part number "MPM3695-25" in the search bar, then click on the part number to navigate to the MPM3595-25's product detail page. Once on the detail page, scroll down to the Design Resources section. Click on "MPSmart Model" to download the MPSmart model.

This should download a zip file containing the model files and associated documentation for the part.

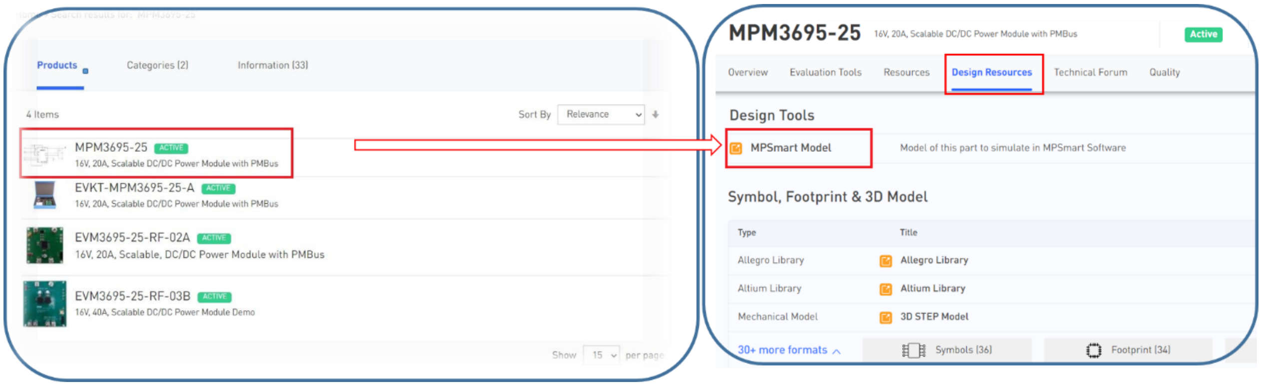


Figure 6: Finding MPSmart Models on the MPS Website

Obtaining Assistance from MPS Support Teams

Another way to obtain the latest IC models is to contact an MPS FAE. They can provide users the required files, and can help users start their design.

The online technical forum is also a good contact window (see the Contacting Technical Support section on page 33).

For further support, send an email to the simulation team at mps-simulation@monotlithicpower.com.

Intranet users can find most of the design resources on the intranet.

BASIC OPERATIONS

This section discusses the basic operations of the MPSmart software, including navigating through the typical application schematics for MPS products, as well as how to create a new project with an MPS IC model.

Navigating Typical Schematics for MPS Products

The MPS simulation team provides various typical application schematics for each MPS IC model to help users start their design. As described in the User Interface Overview section on page 4 and the Selecting and Downloading MPS IC Models on page 7, users can locate the MPS IC models in the pre-installed MPSmart "Examples" folder, download the models from the MPS website, or ask MPS technical support to provide the models.

A typical application schematic for a circuit in Schematic Editor consists of six sections (see Figure 7).

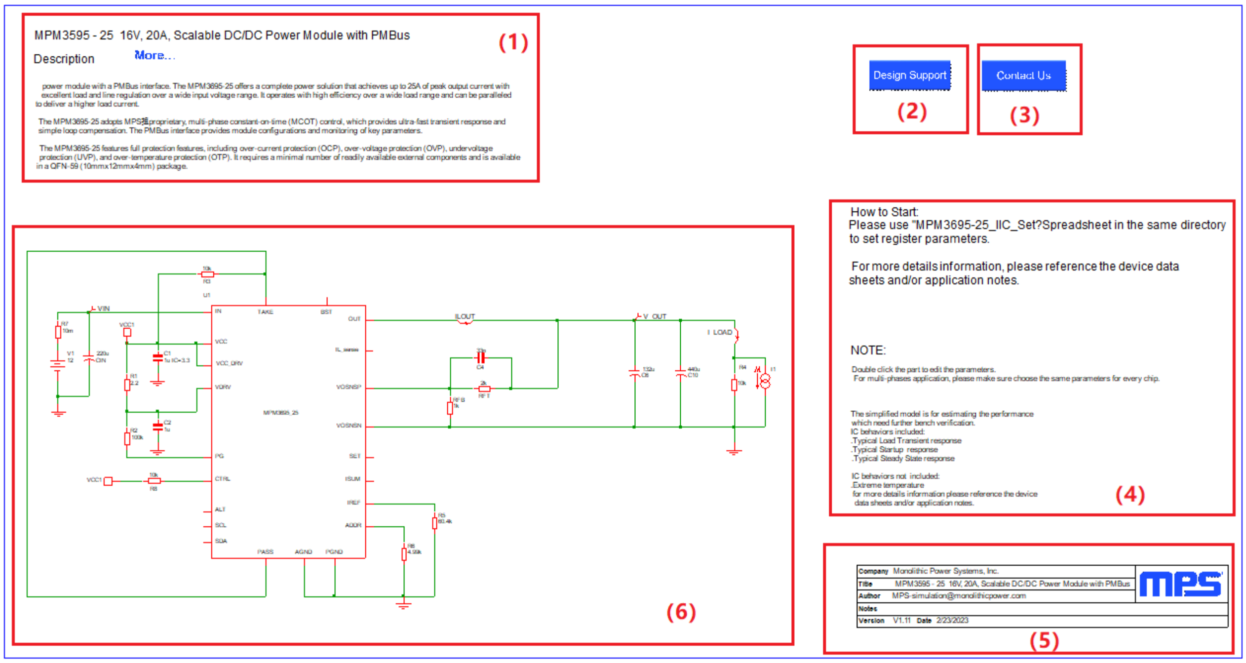


Figure 7: Typical Schematic for an MPS IC Model

1. Description: This section provides a brief introduction to the IC, including basic information such as product specifications, application range, control strategy, and package information. Clicking on "More" opens the product detail page on the MPS website, where users can find comprehensive information on the part (see Figure 8 on page 10).
2. Design Support: This button opens the Design Tool page on the MPS website. Users can explore and use various design tools provided by MPS to facilitate the design process (see Figure 9 on page 10).
3. Contact Us: The "Contact Us" button opens the regional contact information page for technical support and assistance.
4. Model notes: This section provides helpful information about the IC model, such as built-in performance features and how to use any accompanying documentation.
5. Model information: This block displays basic information about the IC model, including the version, date of creation, and author.

6. Typical application schematic: This section provides a fully verified circuit design that can be used for direct simulation to evaluate an application's performance. By adapting and modifying the typical application schematic, users can speed up the design process and test the project using a proven application platform.

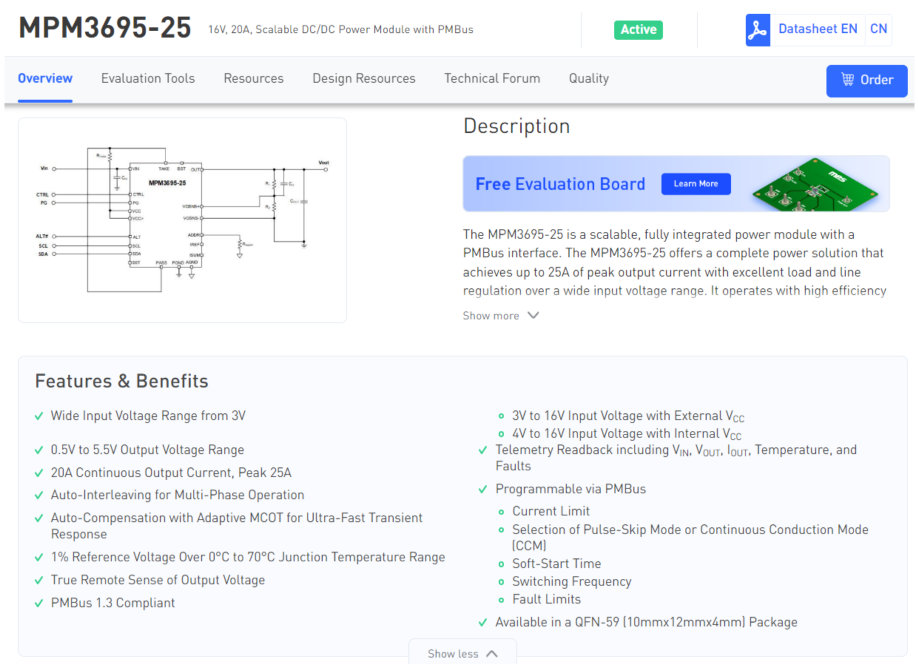


Figure 8: Click 'More' to Open the IC Detail Page

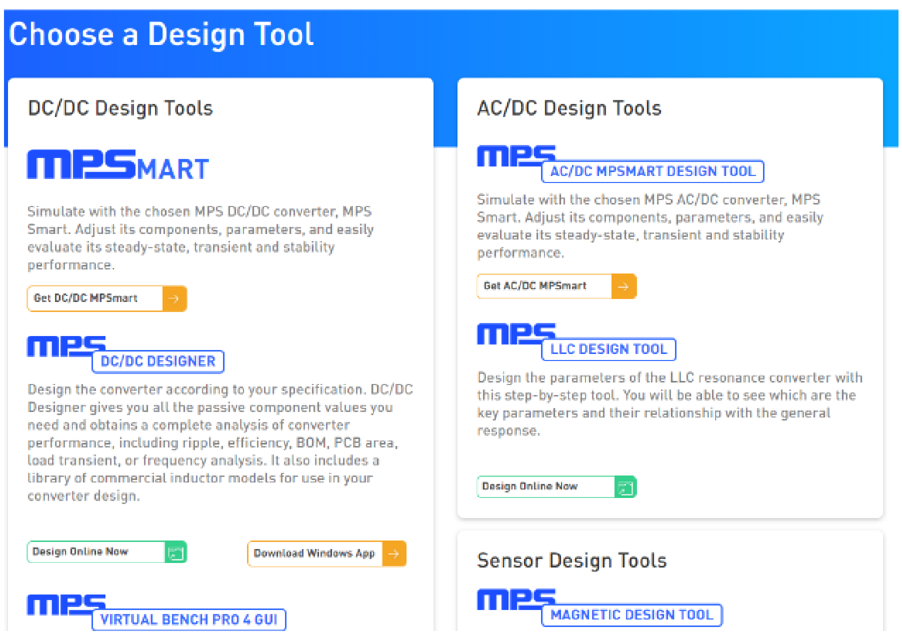


Figure 9: Design Tool Page

Creating a New Project with an MPS IC Model

An easy way to start a new project with an MPS IC model is to implement the design based on the typical application schematic, as discussed in the Navigating Typical Schematics for MPS Products section on page 9. Adapting and modifying the schematic can accelerate the design process.

Another option is to create a new design by clicking "File" in the main menu, and then selecting "New/SIMPLIS Schematic" or "New/SIMetrix Schematic". Save the schematic to the folder. Most MPS IC models are SIMPLIS models, while a few are SIMetrix models.

Select the MPS IC model from the model library to start designing the schematic. Click "Place From Model Library" to add additional components, such as resistors, capacitors, sources, and semiconductor devices (see Figure 10).

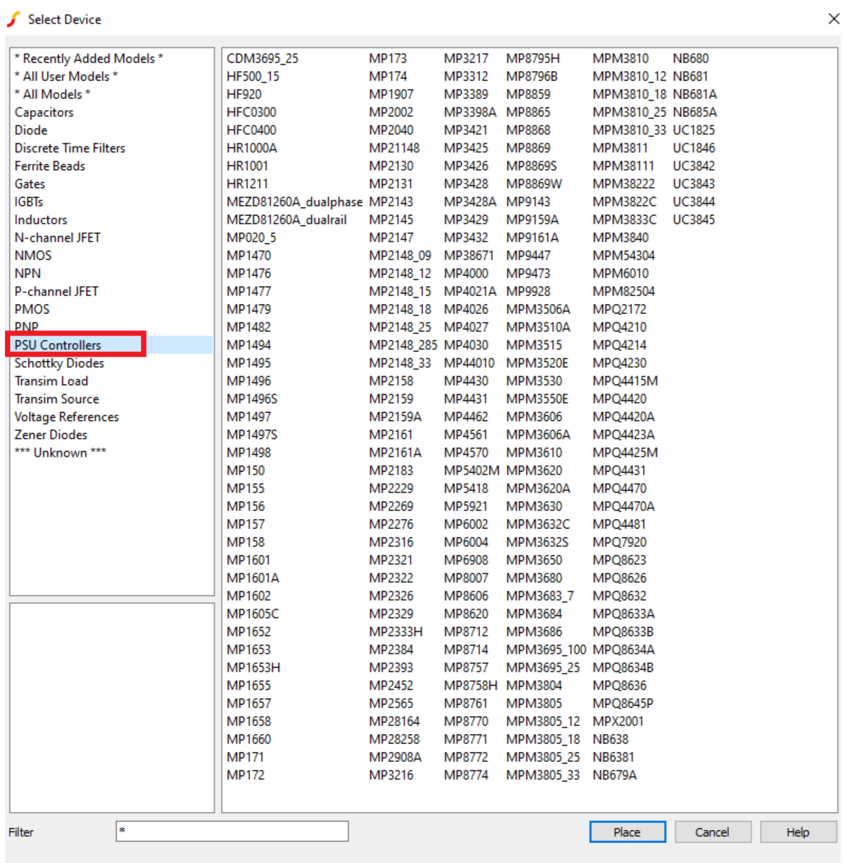


Figure 10: Model Library

Arrange and connect the components in the schematic as required for the design. Now the new project with the MPS IC model is ready for simulation analysis.

In addition to obtaining MPS IC models from the MPSmart model library, users can find more from the MPS website or field support (see the Selecting and Downloading MPS IC Models section on page 7 for more details).

RUNNING A SIMULATION

This section discusses the different types of analysis that can be performed in MPSmart. The available simulations are periodic operating point (POP) simulation, frequency-based AC simulation, and transient simulation.

Periodic Operating Point (POP) Simulation

POP simulation is a specialized transient analysis, which quickly finds the switching steady-state operating point of a circuit. There are two key concepts for POP simulation:

1. POP typically works on a switching circuit.
2. A triggering gate is required for the POP analysis.

POP typically works on a switching circuit. Figure 11 shows the buck circuit with a POP trigger. POP exhibits a periodic switching behavior. POP analysis excels at finding the steady state on/off limit cycle of a stable switching system. This means that POP is successful if the schematic is a stable switching circuit. If not, POP analysis fails.

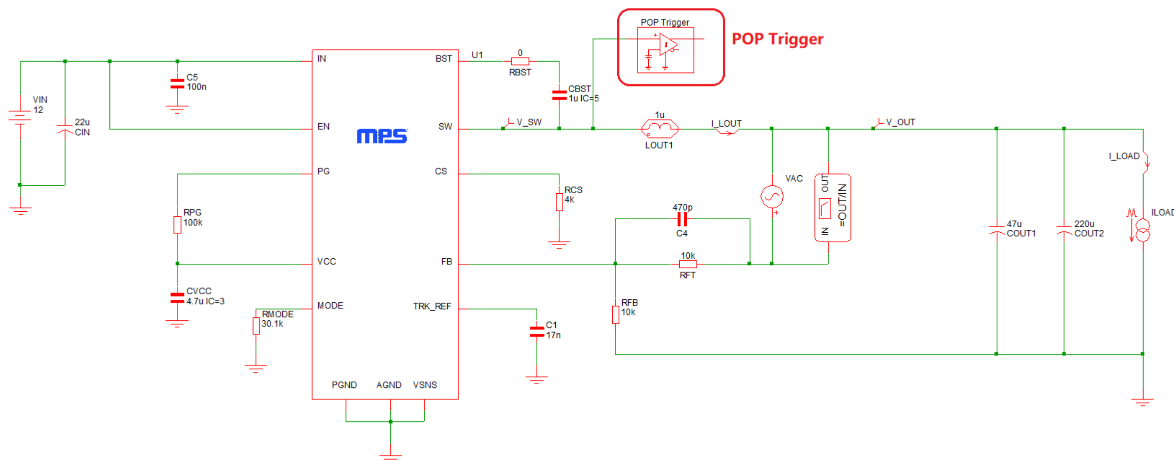


Figure 11: Buck Circuit with POP Trigger

A triggering gate is required for POP analysis. POP analysis requires the user to designate a switching node, which is the least common multiple of the periodic frequencies. The SW pin on the buck circuit can be used to define the periodic frequency (see Figure 11).

The POP Trigger Schematic Device window provides a convenient way to define the trigger gate (see Figure 12). The user can find the device by clicking Menu > Place > Analog Functions > POP Trigger.

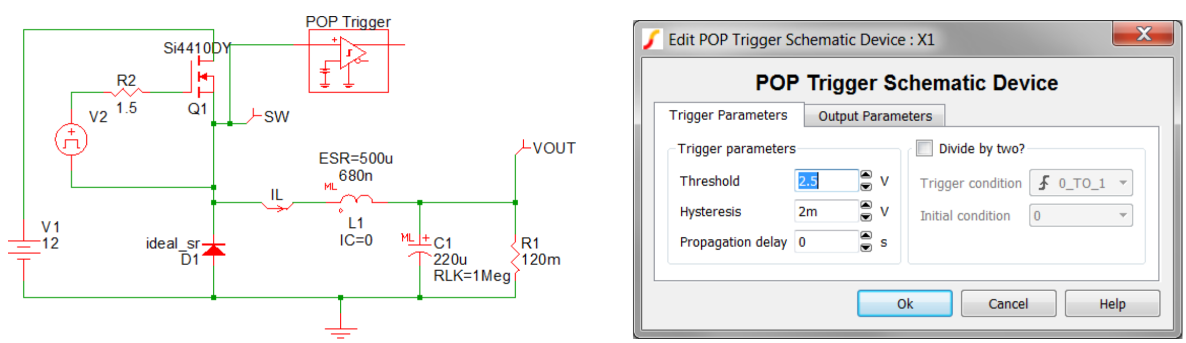


Figure 12: Configure POP Trigger

Set Up POP Analysis

To set up POP analysis, follow the steps below:

1. From the menu bar in Schematic Editor, select Simulator > Choose Analysis
2. Click on the Periodic Operating Point tab.
3. Under the Select Analysis section, check the “POP” box (see Figure 13).
4. The maximum period parameter must be larger than the switching period. Do not set the maximum period parameter to more than two times the switching period.

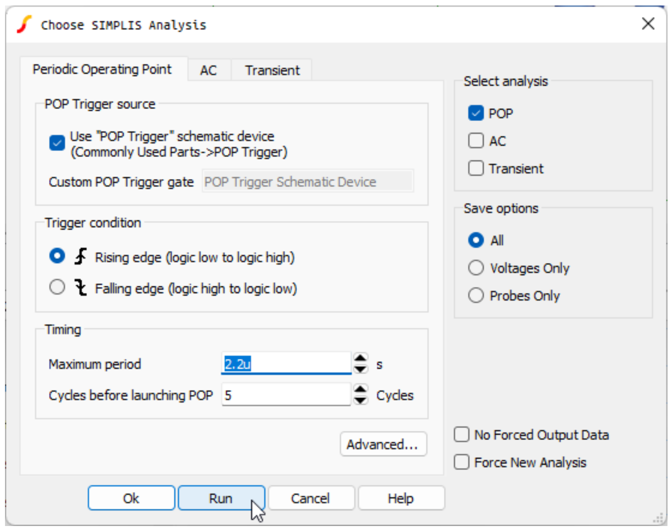


Figure 13: Set Up POP Analysis

5. Once the analysis directive parameters have been entered, click the “Run” button or click “Ok” to save the data. Press F9 to run the simulation. The waveform viewer window will open with the converter’s steady-state waveform. Notice that the converter shows no setting behavior and the waveforms are in perfect steady-state (see Figure 14).

POP analysis helps the circuit reach steady-state faster than running a long transient simulation.

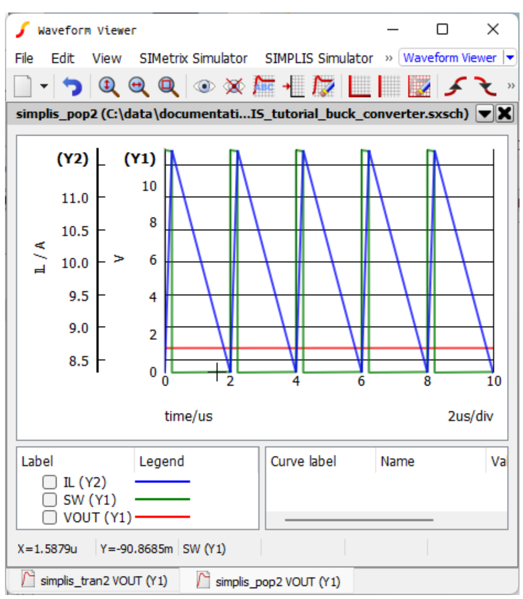


Figure 14: Waveforms for POP analysis

Frequency-Based AC Simulation

AC analysis is used to obtain the frequency response of a switching power supply. AC analysis is performed on the time domain model by first finding the POP, then injecting a single time-domain sinusoidal perturbation signal into the circuit. That means if MPSmart cannot find a stable steady-state periodic operating point, the AC analysis fails. The AC analysis is dependent on POP analysis successfully finding the steady state operating point.

To set up an AC analysis, follow the steps below:

1. Select Simulator > Choose Analysis.
2. Check the “AC” box under the Select Analysis panel. Note that the “POP” box is automatically checked when AC is checked.
3. Select the AC tab at the top to enter the parameters (see Figure 15).

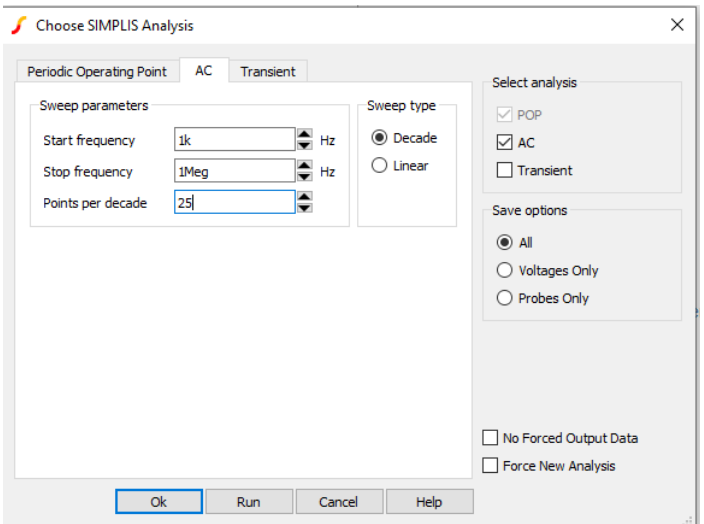


Figure 15: Setting Up AC Analysis

4. Click the “Run” button or click “Ok” to save the data. Press F9 to run the simulation. The waveform viewer window opens with the converter’s frequency response (see Figure 16).

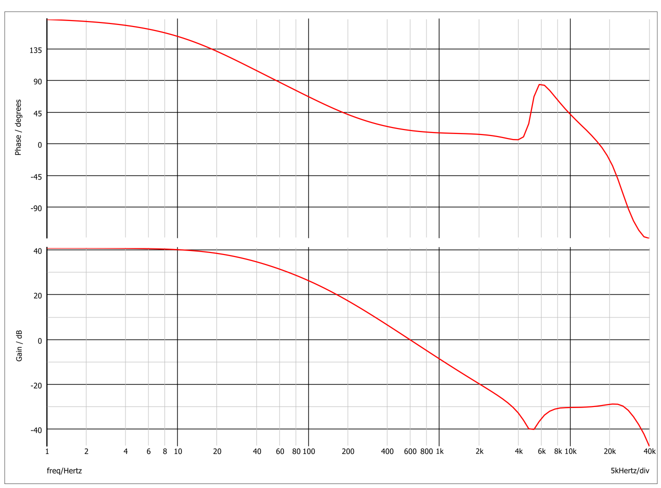


Figure 16: Gain and Phase of AC Analysis

Transient Simulation

Transient simulation is used to analyze the time-domain behavior of a circuit, including its start-up behavior, response to load changes, and protection performance.

Figure 17 shows a transient analysis. To perform a transient simulation, follow the steps below:

1. Under the Simulator menu, select "Analysis".
2. Under the Select Analysis section, check the "Transient" box.
3. Under the Transient tab at the top, enter the parameters as described in the following sections.

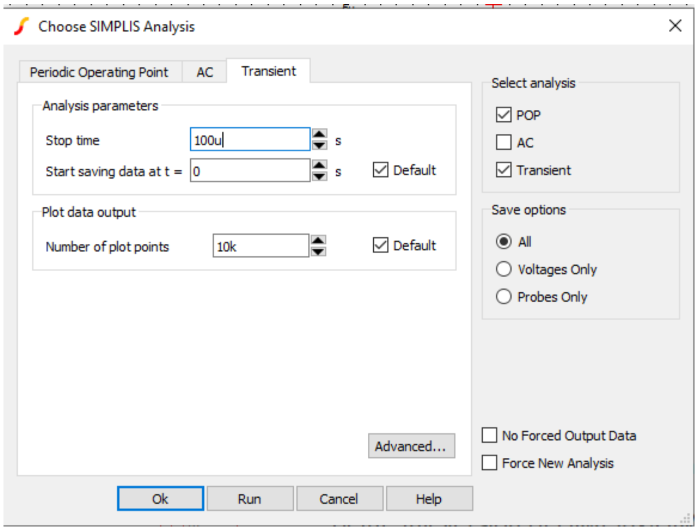


Figure 17: Transient Analysis

4. Click the "Run" button or click "Ok" to save the data. Press F9 to run the simulation. The waveform viewer window opens to show the converter's frequency performance (see Figure 18).

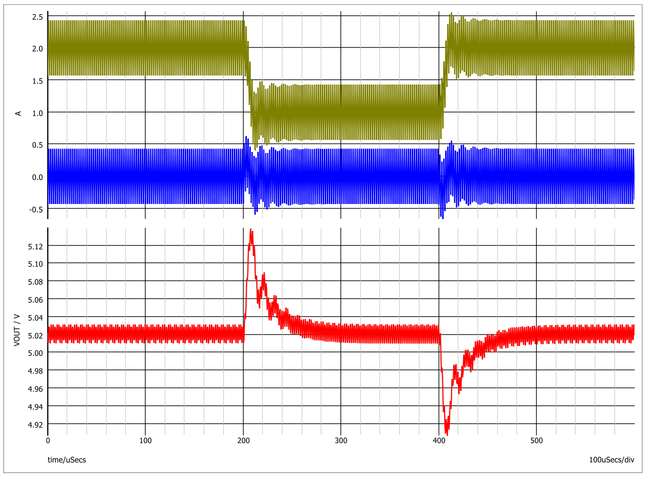


Figure 18: Waveform View of Transient Analysis

By understanding and utilizing these different types of simulations, the user can thoroughly analyze the design's performance and make necessary adjustments to optimize its behavior.

ADVANCED APPLICATIONS

Custom Simulation with MPS Design Assistant

Assistant tools (the advanced applications that work with MPSmart) can be used to assist in circuit design, speed up the design process, and improve the user's experience. This section explores the advanced applications of MPSmart, including custom simulations with the MPS Design Spreadsheet and IIC spreadsheet, as well as designing with the MPS GUI.

Design Spreadsheet Assistant

The Design Spreadsheet Assistant is a helpful tool that automatically calculates essential parameters and values based on the user's input specifications. This includes recommending design, calculating efficiency, power loss, generating a bill of materials (BOM), providing PCB guidelines, designing transformers, and automatically importing design parameters into MPSmart.

To use the Design Spreadsheet Assistant, follow the steps below:

1. Download the required MPSmart (see the Selecting and Downloading MPS IC Models on page 7).
2. Extract the downloaded file, which contains three documents if the model includes design spreadsheet assistant (see Figure 19):

   - <u>Design spreadsheet</u>: This is the main spreadsheet used for design and pre-simulation.
   - <u>Readme</u>: This is a text file containing necessary information and comments.
   - <u>Document folder</u>: This folder contains MPSmart schematics and necessary accessory files.

Ensure that the design spreadsheet and document folder are located in the same path in case of a file call error.

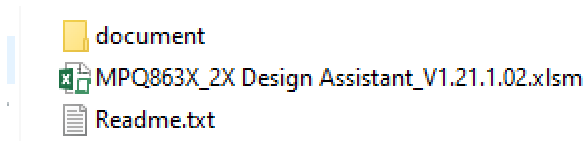


Figure 19: Design Spreadsheet for MPSmart

3. Open the spreadsheet. There are three sheets in the design spreadsheet. The instruction sheet is a brief guide to the steps and basic information about the design spreadsheet (see Figure 20 on page 17).

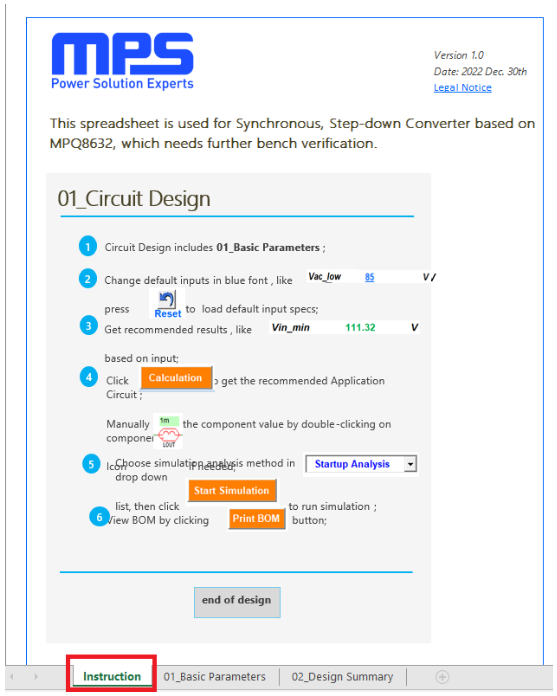


Figure 20: Instruction Sheet

4. In the "01_Basic Parameters" sheet, input the design specifications. Choose an IC part number. After inputting the specifications, users can obtain power loss data, efficiency curves, a BOM, and PCB guidelines (see Figure 21).

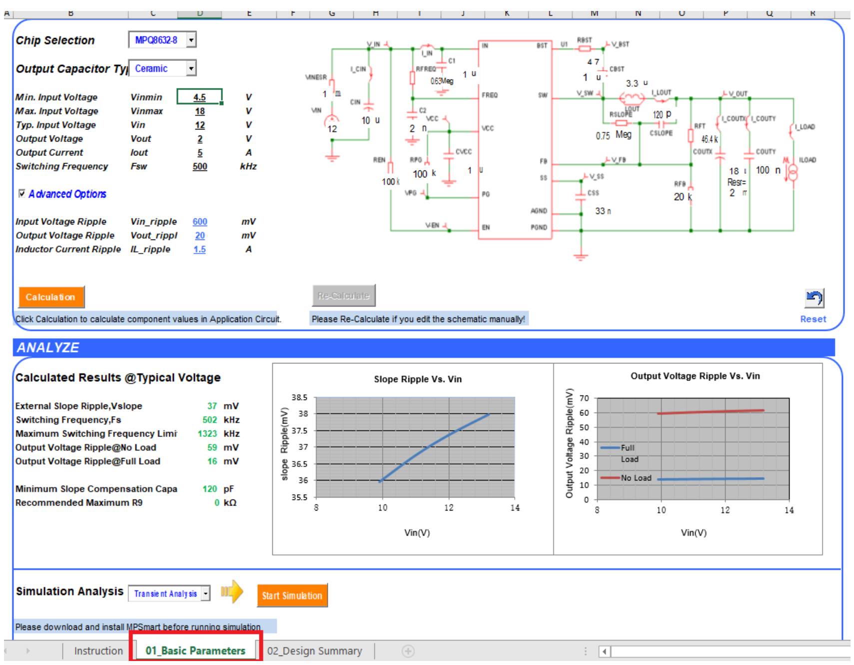


Figure 21: Basic Parameters Sheet

5. The "02_Design Summary" sheet has all the related information. The "01_Basic Parameters" sheet is listed in this sheet. The user can obtain a completed report about the customized circuit (see Figure 22 on page 18).

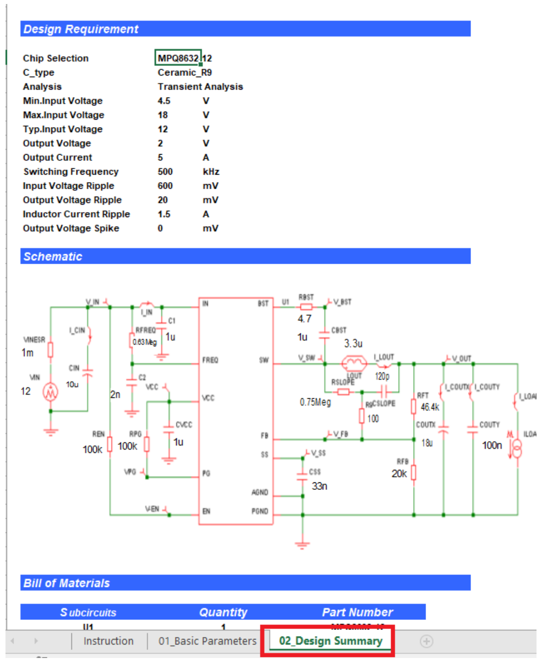


Figure 22: Design Summary Sheet

6. Another important feature is that users can edit and modify the design circuit in the “01_Basic_Parameters” sheet (see Figure 21 on page 17). Select “Analysis Type” in the design sheet, then click the “Start Simulation” button to automatically start and load the customized design circuit (see Figure 23). Press F9 to run and check the performance.

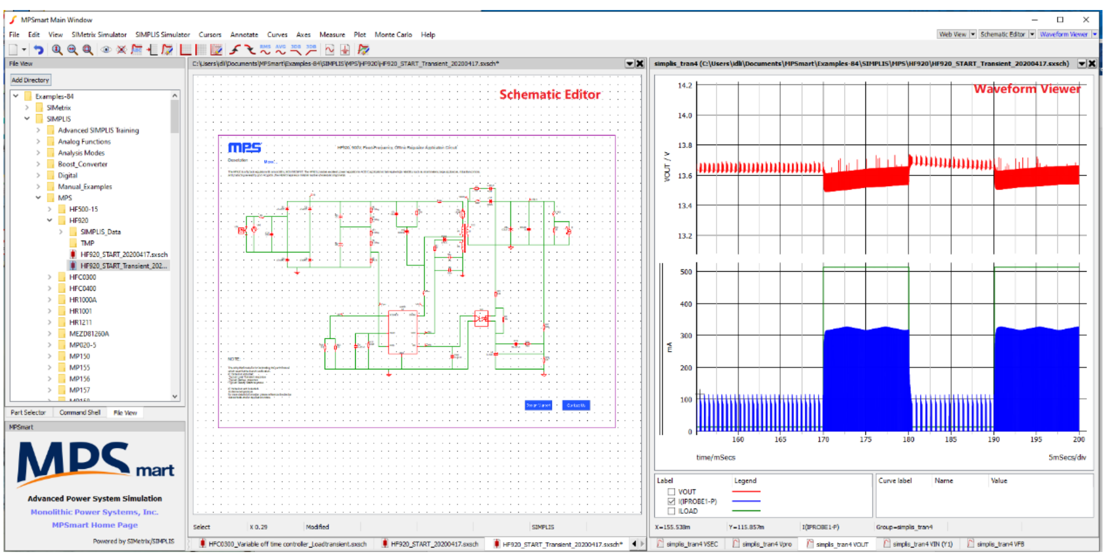


Figure 23: Load Design and Run Simulation in MPSmart with Design Spreadsheet

By using the Design Spreadsheet Assistant, a user can save time during the design process, as well as ensure that the design meets the specifications.

IIC Spreadsheet for Simulation

Similar to the Design Spreadsheet Assistant, the IIC spreadsheet is a convenient tool that simplifies the process of selecting the appropriate parameters without knowing the specific IIC addresses or values. This helps streamline the design process, ensuring the simulations are accurate and efficient.

After downloading the MPSmart model with the IIC spreadsheet assistant, follow the steps below:

1. Extract the download file, which usually contains three types of documents (see Figure 24).
    - IIC spreadsheet: This spreadsheet is used to set the IIC parameters and pass them to the MPSmart model.
    - MPSmart Schematic: This is the MPSmart target schematic (e.g. MPQ7225_Schematic.sxsch) (see Figure 24).
    - Document folder: This folder contains a text file with all the parameters passed.

Ensure the design spreadsheet and document folder are located in the same path in case of a file call error.

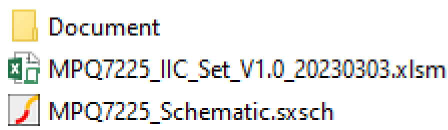


Figure 24: Example of IIC Spreadsheet Application

2. Open the IIC spreadsheet. The introduction page gives a brief step-by-step guide on how to use the spreadsheet to achieve the target (see Figure 25).

Figure 25: Introduction Sheet of the IIC Spreadsheet

3. Enter the design specifications in the appropriate fields in the "IIC setup" sheet (see Figure 27 on page 20). Click the "Save Now" button. A text file containing all of the setting parameters should be saved in the document folder (see Figure 26).

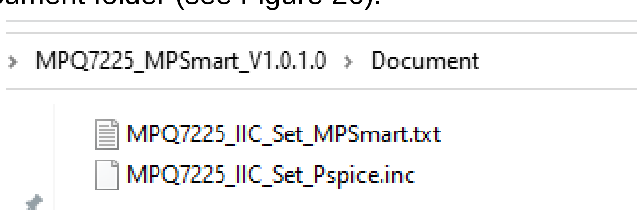
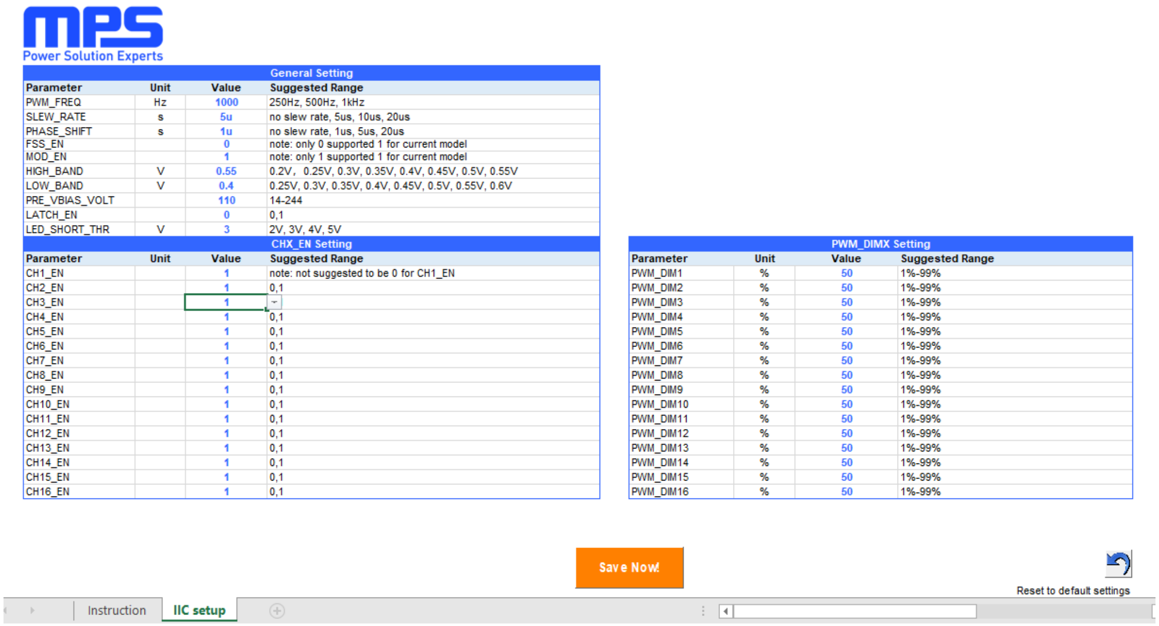


Figure 26: Pass Parameters File Created by the IIC Spreadsheet

In this example, "MPQ7225_IIC_Set_MPSmart.txt" is used to pass parameters to the MPSmart model. "MPQ7225_IIC_Set_Pspice.inc" is used to transfer parameters to the Pspice model. Click the "Reset to default setting" button to reset all fields in the spreadsheet to their default values.



General Setting			
Parameter	Unit	Value	Suggested Range
PWM_FREQ	Hz	1000	250Hz, 500Hz, 1kHz
SLEW_RATE	s	5u	no slew rate, 5us, 10us, 20us
PHASE_SHIFT	s	1u	no slew rate, 1us, 5us, 20us
FSS_EN		0	note: only 0 supported 1 for current model
MOD_EN		1	note: only 1 supported 1 for current model
HIGH_BAND	V	0.55	0.2V, 0.25V, 0.3V, 0.35V, 0.4V, 0.45V, 0.5V, 0.55V
LOW_BAND	V	0.4	0.25V, 0.3V, 0.35V, 0.4V, 0.45V, 0.5V, 0.55V, 0.6V
PRE_VBIAS_VOLT		110	14-244
LATCH_EN		0	0,1
LED_SHORT_THR	V	3	2V, 3V, 4V, 5V

CHX_EN Setting			
Parameter	Unit	Value	Suggested Range
CH1_EN		1	note: not suggested to be 0 for CH1_EN
CH2_EN		1	0,1
CH3_EN		1	
CH4_EN		1	0,1
CH5_EN		1	0,1
CH6_EN		1	0,1
CH7_EN		1	0,1
CH8_EN		1	0,1
CH9_EN		1	0,1
CH10_EN		1	0,1
CH11_EN		1	0,1
CH12_EN		1	0,1
CH13_EN		1	0,1
CH14_EN		1	0,1
CH15_EN		1	0,1
CH16_EN		1	0,1

PWM_DIMX Setting			
Parameter	Unit	Value	Suggested Range
PWM_DIM1	%	50	1%-99%
PWM_DIM2	%	50	1%-99%
PWM_DIM3	%	50	1%-99%
PWM_DIM4	%	50	1%-99%
PWM_DIM5	%	50	1%-99%
PWM_DIM6	%	50	1%-99%
PWM_DIM7	%	50	1%-99%
PWM_DIM8	%	50	1%-99%
PWM_DIM9	%	50	1%-99%
PWM_DIM10	%	50	1%-99%
PWM_DIM11	%	50	1%-99%
PWM_DIM12	%	50	1%-99%
PWM_DIM13	%	50	1%-99%
PWM_DIM14	%	50	1%-99%
PWM_DIM15	%	50	1%-99%
PWM_DIM16	%	50	1%-99%

Figure 27: IIC Set-Up Sheet

4. Open the MPSmart schematic. Click Tools > Load Component Values > Select a File to Load (see Figure 28). Select and open the text file named "MPQ7225_IIC_Set_MPSmart.txt" in the document folder. Click "Ok" to complete the parameters transfer. Each time the setting parameters are modified in the IIC spreadsheet, repeat steps 3 and 4 to reload the parameters to the MPSmart schematic.

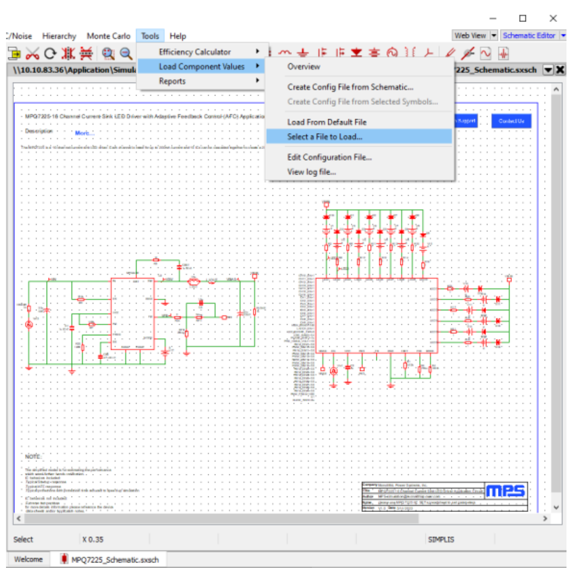


Figure 28: Load the IIC Spreadsheet Parameters

5. Press F9 in MPSmart to run the simulation and start to explore the design.

By using the IIC spreadsheet for simulation, users can simplify the process of parameter selection and transferring for simulations, ensuring accurate results and speeding up the design process.

Design with the MPS GUI

MPSmart can import the configuration file generated by the MPS graphical user interface (GUI) to improve the design process and facilitate real-time interaction with the IC models.

The GUIs mentioned here are designed for to display digital products, configure register parameters, and read/write IC chips. The register parameters in the GUI can also be exported to be used in the MPSmart model. Configure the register data in the GUI, and the same data can be utilized in MPSmart and the real chip. This is convenient for chips that have a large number of parameters to be configured and passed to the simulation.

Users can input the parameters in the configuration page (see Figure 29). Skip this step to use the default values.

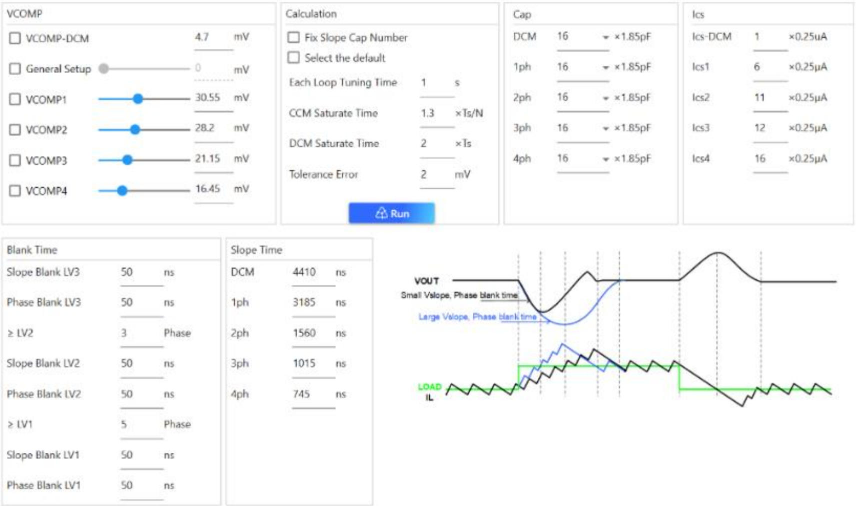


Figure 29: Configure Page in the GUI

When all the required parameters are ready, click the “Export” button to save the data as a text file that MPSmart/SIMPLIS can recognize (see Figure 30).

Similar to the IIC spreadsheet application (see Figure 29), open MPSmart and load the saved configuration file. The simulation circuit should load the data from the GUI, and then the simulation circuit should be ready to run.

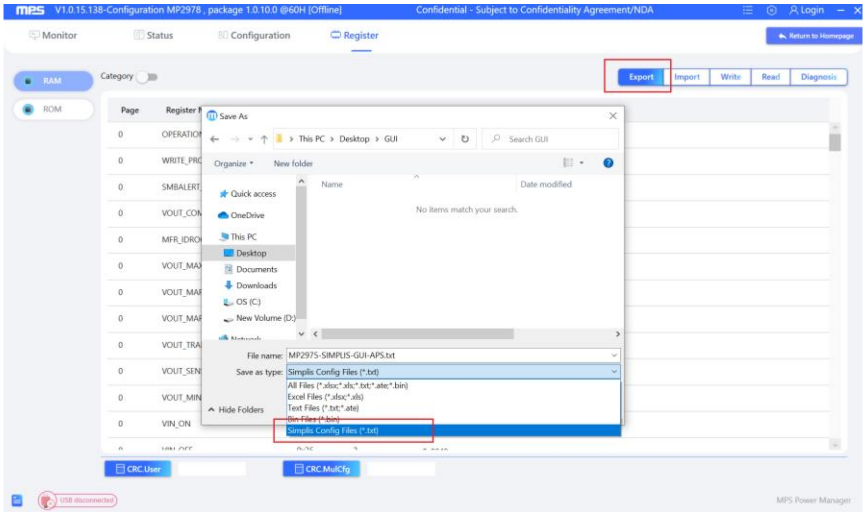


Figure 30: Export the Parameters for MPSmart from the GUI

This method is similar to the IIC spreadsheet, but is more suitable for applications with a large number of parameters that need to be imported into MPSmart. The GUI gives users easy control over the design and simplifies the process of passing parameters to the simulation.

Design by Product

MPS has developed over 300 MPSmart models with a wide range of products. These models cover a variety of categories, including AC/DC products, DC/DC converters, power modules, automotive products, analog circuits, LDOs, LED converters, and multi-phase controllers. Each model comes with fully tested and operational design examples.

MPS is continuously working to develop more product models and improve the modeling capabilities. This section shows a selection of typical examples.

MP6005: Flyback/Forward Controller with Primary-Side Regulation (PSR) and Secondary-Side Regulation (SSR)

The MP6005 is specifically designed as isolated solution with primary-side regulation (PSR) for flyback applications and secondary-side regulation (SSR) for forward applications. Both applications are created using MPSmart. The MP6005 can also use SSR in a flyback topology.

Figure 31 shows the verified typical application circuit (both a flyback converter with PSR and a forward controller with SSR are available). Frequency dithering, overload protection (OLP), and over-voltage protection (OVP) are also modeled for the MP6005 in MPSmart to support the full functions simulation.

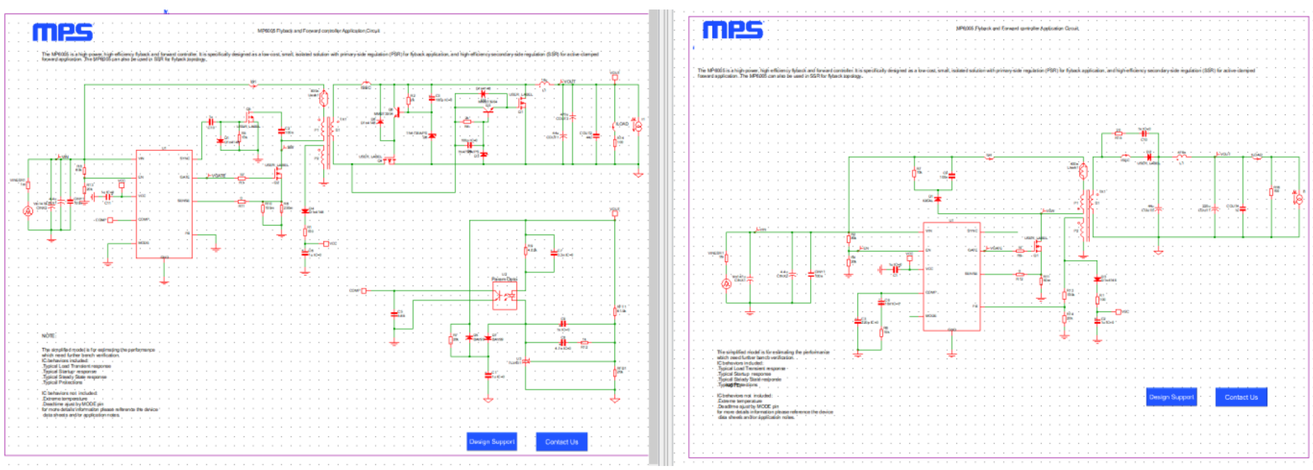


Figure 31: MP6005 Schematic of Forward Application with SSR and Flyback Application with PSR

Figure 32 on page 23 shows the load transient behavior of both flyback and forward converters separately. Users can also explore the line regulation, steady state performance, input startup, input shutdown, OVP behavior, and short-circuit protection (SCP) behavior by varying the test conditions with the model.

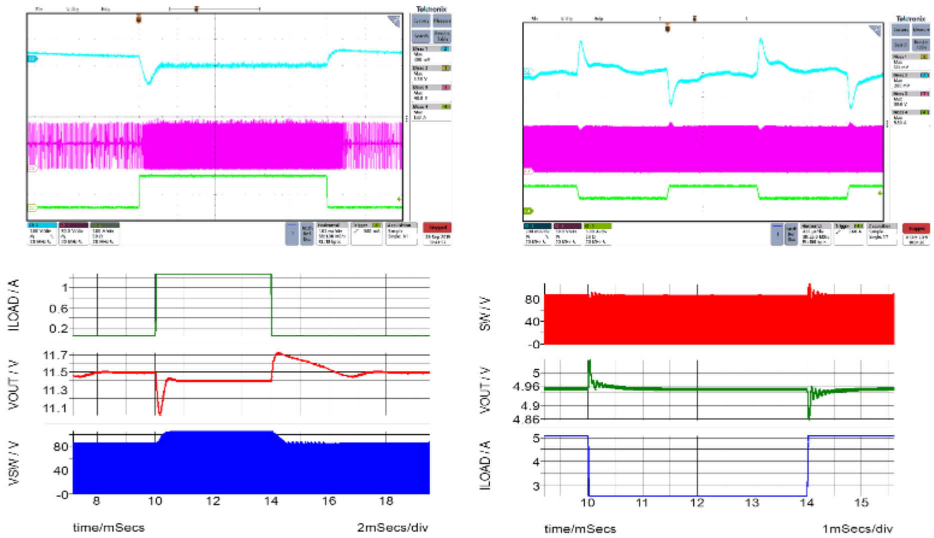


Figure 32: Flyback Application with PSR and Forward Application with SSR Simulation Waveform on the MP6005 Model

With this type of application, users can easily explore complex designs under different operating conditions.

MPM3695-25: Scalable, DC/DC Power Module

The MPM3695-25 provides a power solution that achieves excellent load and line regulation across a wide input voltage (V_{IN}) range. Single-phase and multi-phase applications are presented in MPSmart.

Figure 33 shows a single-phase example. This typical application helps users start the design quickly. Users can run the simulation directly, modify the parameters to verify the design, or use a partial schematic to start a new design. Double click the IC in the schematic to enter the parameters.

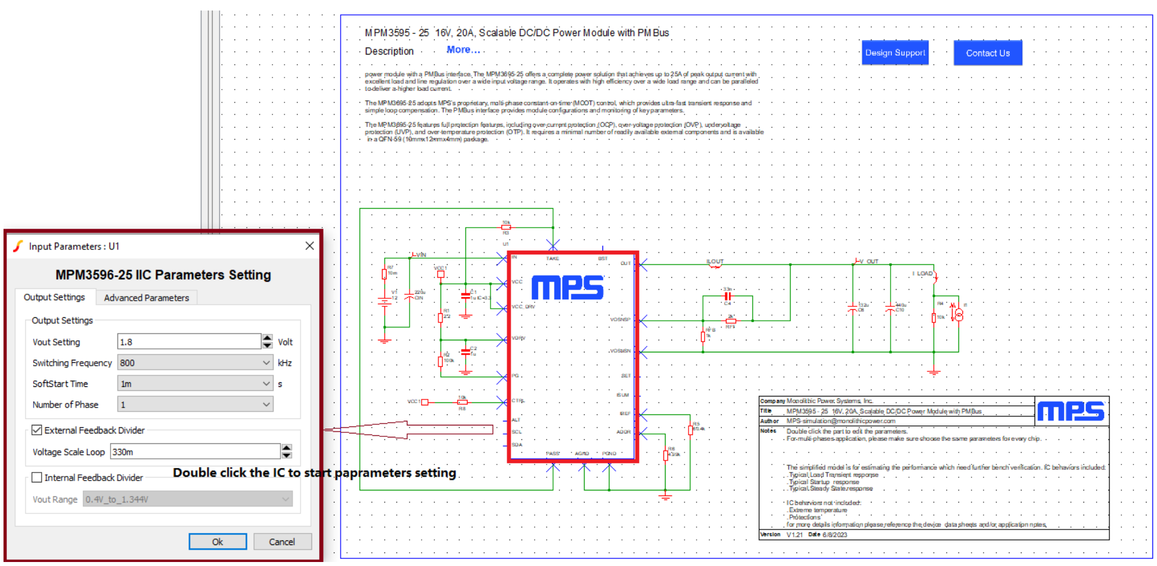


Figure 33: MPM3695-25 MPSmart Schematic for Single-Phase Control

The constant-on-time (COT) control and ramp compensation scheme are well modeled to ensure model reliability and accuracy.

Figure 34 shows comparative results for users to evaluate the behavior of the power module.

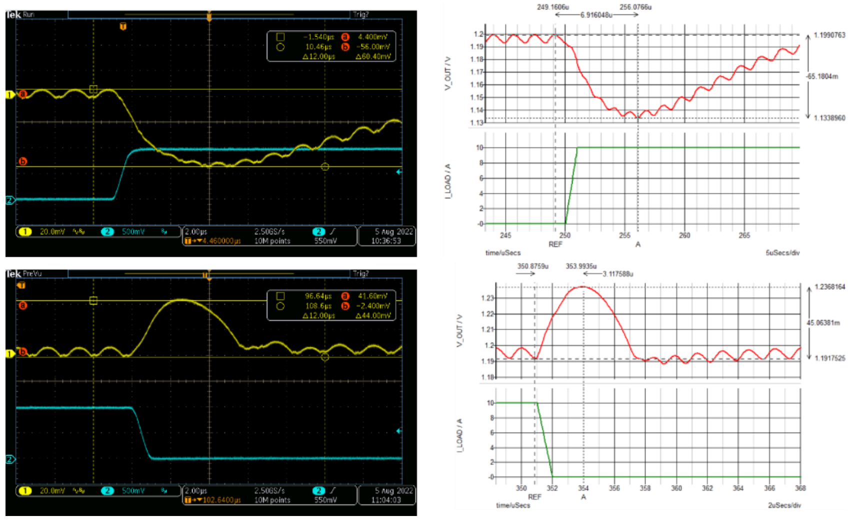


Figure 34: MPM3695-25 MPSmart (Bench vs. Simulation)

Figure 35 shows a four-phase application of the MPM3695-25 power module. The master and slave configuration performs master and slave IC detection, communication between master and slave ICs, and COT loop control . This example provides insight into the performance of a multi-phase power module set-up.

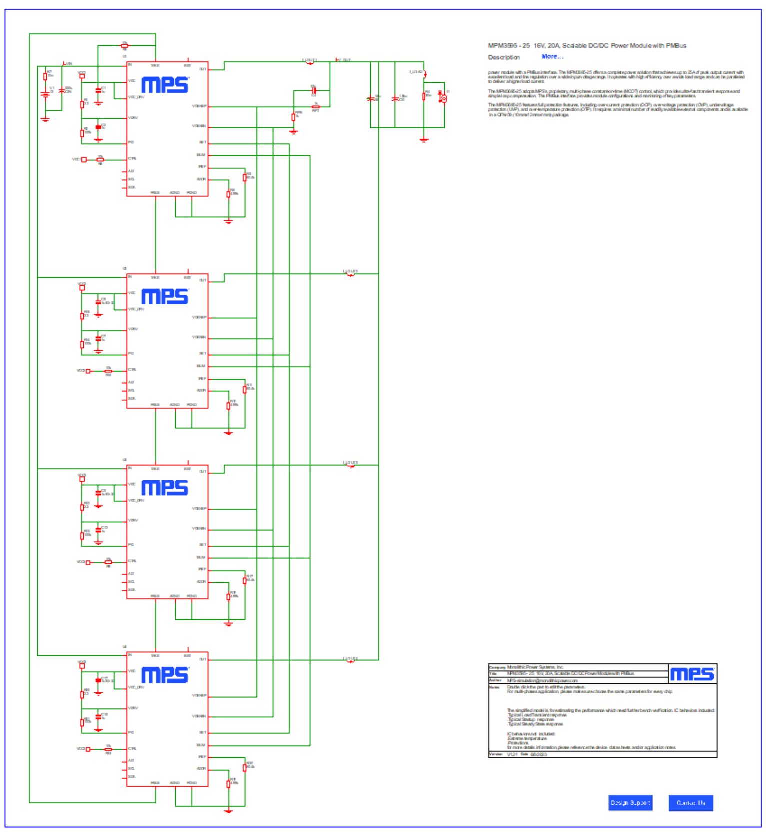


Figure 35: MPM3695-25 MPSmart (4-Phase Configuration)

Design by Application

LLC Control with Synchronous Rectification (SR)

Figure 36 shows an LLC converter with a CC-CV control power supply solution to ensure the target output works properly. Instead of diodes, synchronous rectification (SR) is used to reduce the voltage drop.

Two MPSmart models of two devices are used in this application. The first is an LLC controller with current mode control. It is designed to operate at frequencies up to 500kHz during steady state, with the burst/skip mode switching at light loads and adaptive dead time adjustment with minimum and maximum limits.

The other device model is a dual, fast turn-off, intelligent rectifier. This IC model drives two MOSFETs and regulates their forward voltage drop to improve the efficiency and ensure the turn-on/off speed.

The simulation takes about 5 minutes while running the combination simulation with the proper settings. Users can observe the steady-state waveform (see Figure 36).

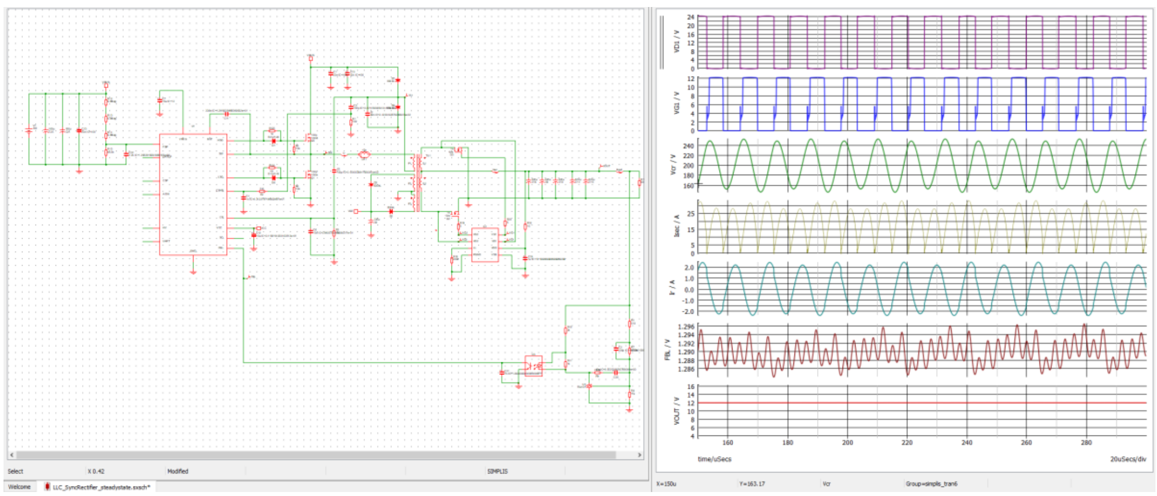


Figure 36: LLC Controller with Synchronous Rectification

Virtual EMI Lab Simulation Model

This example describes an approach to EMI simulation through MPSmart. Users typically perform an EMI test after hardware is complete. This can be time consuming and costly if major modifications are required.

A virtual lab is designed to assist users to better understand EMI design principles and to identify potential EMI problems during product design. By focusing the simulation on EMI, users can identify the main factor of EMI performance, and find possible solutions before hardware is even built.

Figure 37 on page 26 shows the main elements of a conducted emissions (CE) model in the virtual simulation lab.

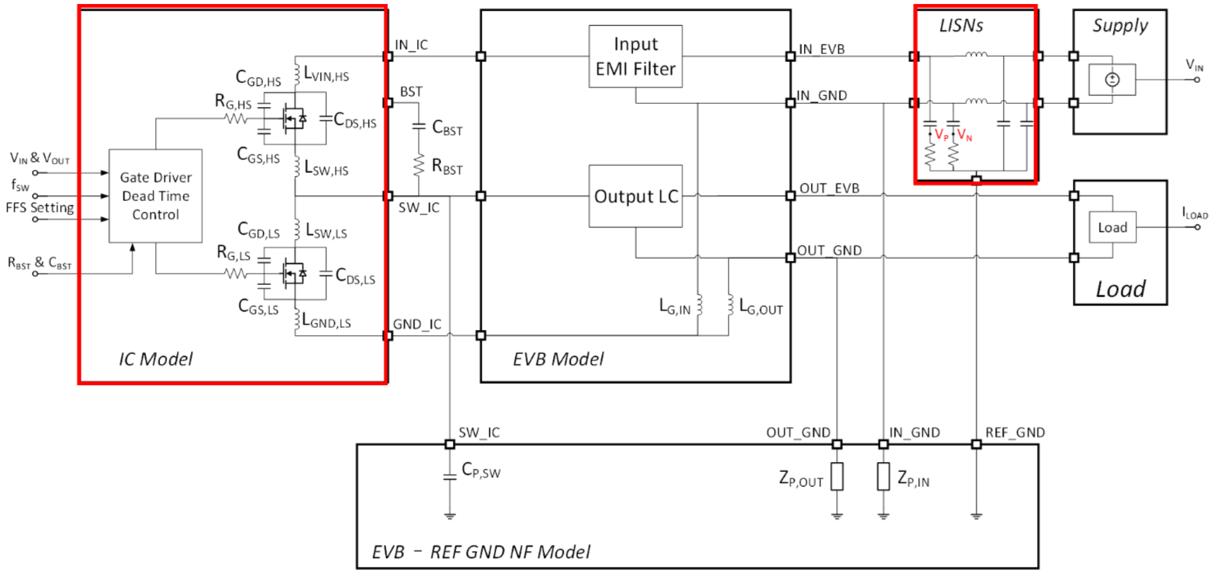


Figure 37: Diagram of Conducted Emissions (CE) Model

IC model: This IC model is built to obtain actual and accurate switching waveforms for further analysis. The basic functions cover the items listed below:

- Basic functions of chip, such as input, output, switching frequency (f_{SW}), and the frequency spread spectrum (FSS) setting
- Switching converter (open-loop converter is used to reduce the simulation complexity)
- Switch modeling, MOSFETs and package parasitic model, and detailed gate driver modeling

EVB model: This block is built for parasitic extraction of the PCB and passive component impedance extraction, described below:

- PCB parameters extraction for key capacitance/inductance matrix of traces and loops
- Passive component impedance extraction (data is used to build a database for typical capacitors and inductors)
- The line impedance stabilization network (LISN) block is used to measure line and neutral EMI noise

In addition to the modeling work for CE simulation, the inductor near E-field model and cable NF coupling model can also be considered for further RE simulation.

Figure 38 on page 27 shows the top-level MPSmart schematic, including the IC models, EMI filter, LISN, output cable model, and associated impedance network models.

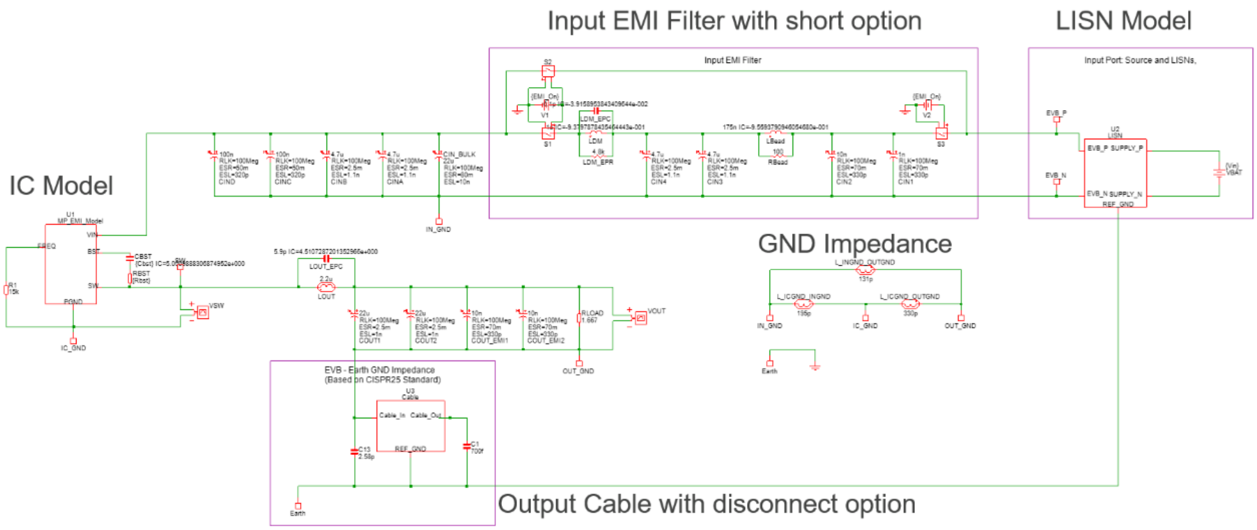


Figure 38: Top-Level MPSmart Schematic

With proper modeling and simulation working for each sub-model block, the simulation result obtained matches the bench measurement (see Figure 39).

The simulation result shows higher background noise. In addition, the current simulation result is based on FFT without considering the EMI detector's influence. Therefore, it can only be used to estimate the peak noise. The next step would be to improve the resolution to optimize background noise and improve the virtual lab for more applications.

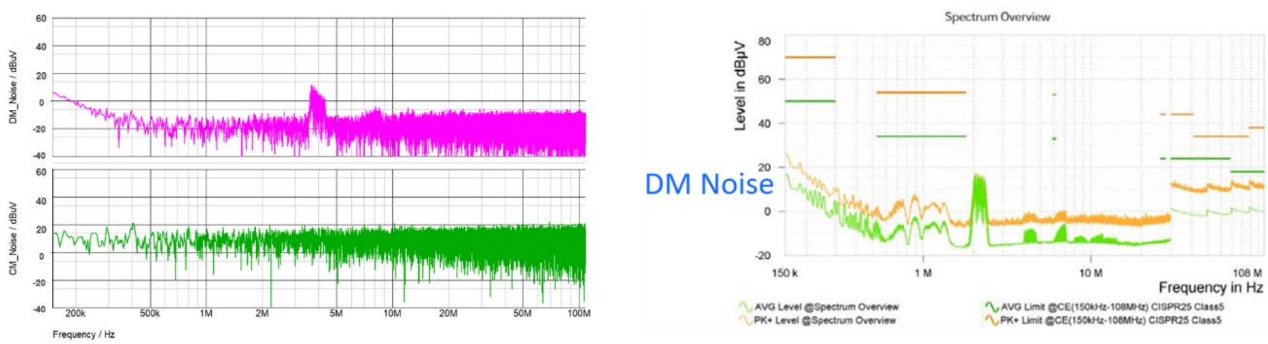


Figure 39: Virtual Lab Result vs. Measurement

In this section, examples based on MPS products or applications are given to show the simulation capability of MPSmart. Users can easily verify MPS products directly, address concerns in the design process, or explore new design and ideas with comprehensive simulation examples.

Additional Resources

In addition to the MPSmart models, MPS provides a wide range of design tools, including the SPICE type models, tools, and online software to help users select the proper IC, as well as tools to help design the application, BOM, and analyze the design.

- DCDC Designer (https://www.monolithicpower.com/en/design-tools/design-tools/dc-dc-designer-online.html): DC/DC Designer is a free power design tool that takes basic input and output specifications, quickly provides a full schematic, smart design, basic waveforms viewing, BOM, bode plot, and efficiency evaluation for DC/DC converters. Online and Windows desktop versions are available.

- Spreadsheet tools: Spreadsheet tools are popular tools used in various applications. They are widely used for transformer design, in-depth calculation for efficiency and smart parameters recommendation. They are also designed to assist with simulation models mentioned in the Custom Simulation with MPS Design Assistant section on page 16. Download the spreadsheet tools from each product page on the MPS website.
- Pspice model: As a widely accepted SPICE-type model, MPS provides Pspice models with completed schematics, as well as models that are fully compared and verified with the bench result. The reliable applications help users quickly start a design and see the performance of MPS products. Download the model from each product page on the MPS website.
- Other design tools (https://www.monolithicpower.com/en/design-tools/design-tools.html): Explore the MPS website for more useful tools, such as the inductor selector, LLC design tool, and magnetic design tool.

MPS continues to focus on developing and improving the product library to expand the comprehensive design applications for users to conveniently evaluate MPS power solutions, and to optimize their designs accordingly.

TROUBLESHOOTING AND SUPPORT

This sections covers common issues that users may encounter when working with MPSmart, and provides guidance on how to contact technical support for further assistance.

Common Issues and Solutions

Below are some common issues that users may experience when working with MPSmart and the possible solutions:

Q1: MPSmart does not launch or crashes unexpectedly.

A: MPSmart is updated at least once a year. Ensure that the latest version of the software is installed. If the issue persists, try to download the latest version (https://www.monolithicpower.com/en/design-tools.html) and reinstall MPSmart.

Q2: What is the difference between MPSmart and SIMetrix/SIMPLIS?

A: MPS distributes the MPSmart simulation tool under license from SIMPLIS Technologies. It is a free downloadable version that offers full schematic capture and waveform viewing/analysis capability. All features are enabled for MPSmart models. As for third-party SIMPLIS or SIMetrix models, users must install the fully licensed SIMetrix/SIMPLIS.

Q3: Can the MPSmart IC model be converted for use in SPICE?

A: No, MPSmart IC models cannot be used directly in a SPICE simulator. Currently, some semiconductor devices, SPICE MOSFETs, diodes, BJTs, IGBTs, JFETs, and Schottky/Zener diodes are supported in MPSmart.

In addition to the MPSmart model, MPS also provides SPICE-type models. Users can download these models from the MPS website.

Q4: An error occurs while running the MPSmart IC model.

A: If the following error occurs when running an MPSmart model (see Figure 40), try switching the simulator. Select the "Simulator/Switch to SIMPLIS Mode" menu.

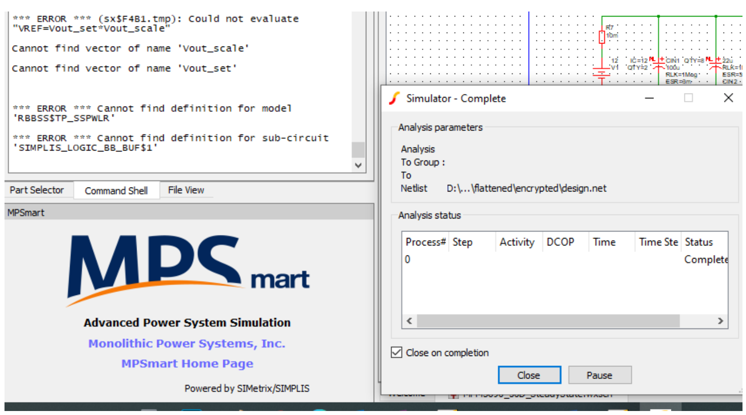


Figure 40: Selecting Wrong Simulator Error

Q5: Simulation failed in POP analysis.

A: There are many reasons for a POP analysis failure. Circuit instability is often the main reason. To solve the issue, perform a time-domain transient analysis before starting POP analysis. This important step can help identify the issue within the circuit and why it might be unstable.

If the simulation waveforms do not work properly, this indicates that the circuit is unstable, which can cause POP analysis to fail.

Figure 41 shows that V_{OUT} has a low-frequency oscillation. POP analysis fails in this situation. Note that "stable" means that the circuit state variables have the same value at any given point in every single switching cycle.

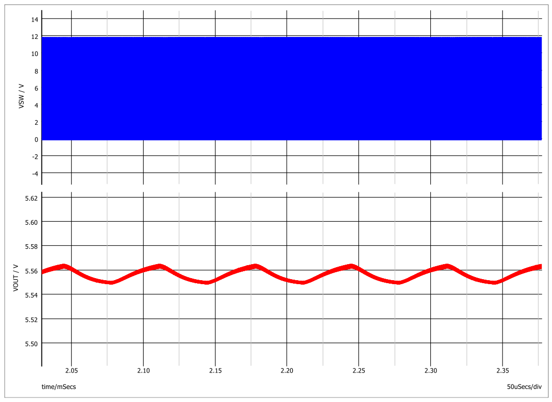


Figure 41: POP Analysis Fails If the Circuit Is Not Stable

AC analysis is completely dependent on whether the steady operating point can be found during the POP analysis (see Running a Simulation on page 12). If POP analysis fails, the AC analysis is not performed. A warning message appears in the command shell.

Q6: What causes the MPSmart simulation to slow down?

A: There are many factors that cause the MPSmart simulation to slow down. A common cause is the complexity of the circuit, such as in multi-phase simulations. It is important to evaluate the simulation goal. Focus on the elements that contribute to the simulation goal. Consider the following steps to improve the simulation:

1. Disable extra functions: Many models provide a range of features, including soft start (SS), steady state, and protection functions. Using all these features at the same time can result in slower simulation speeds in a complicated circuit. Consider disabling functions that are not being used for the current simulation.
2. Delete unused phases: In multi-phase applications, if there are phases that do not contribute to the current simulation, consider deleting or disabling them. This can greatly reduce the complexity of the simulation and help speed it up.
3. Simplify a circuit by ignoring the parasitic parameters: Parasitic parameters play an important role in obtaining accurate results in simulations. They can also increase the complexity of the circuit, which may slow down the simulation speed and cause convergence issues. In many cases, there is a trade-off between simulation speed and accuracy.

Q7: "The number of components has exceeded the limit…" Error

A: Figure 42 shows an error that can occur when running a simulation with an MPS IC model.

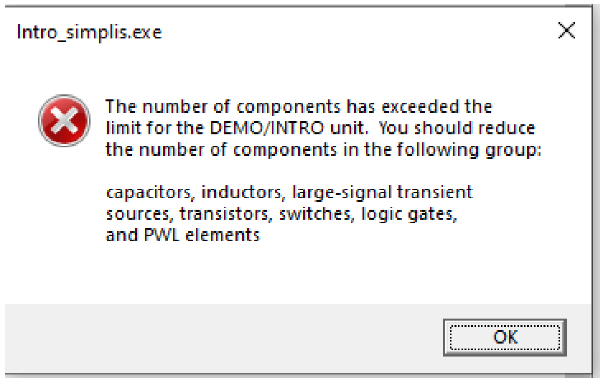


Figure 42: POP Analysis Fails If the Circuit Is Not Stable

Ensure that the latest model is being used and that the latest MPSmart software is installed. If it still does not work, email mps-simulation@monolithicpower.com to request a circuit upgrade.

Q8: Simulation results on the MPSmart model are not as expected or contain errors.

A: Double-check the schematic for any incorrect connections, component values, or parameter settings. Ensure that the correct simulation settings are applied, and confirm whether the MPS IC model is up to date. Users can use the typical schematics provided by MPS as a design reference.

Q9: Difficulty locating a specific MPS IC model or schematic.

A: See the Selecting and Downloading MPS IC Models section on page 7 for detailed instructions on how to locate and download MPS IC models and schematics. If the desired IC model still cannot be found, contact technical support for assistance.

Q10: MPSmart models version information check.

A: The name of the downloaded zip folder contains version information. Figure 43 shows an MPSmart model for the MP2192C, version 1.0. Check the typical model schematic provided by MPS. This also shows the version number and more detailed information.

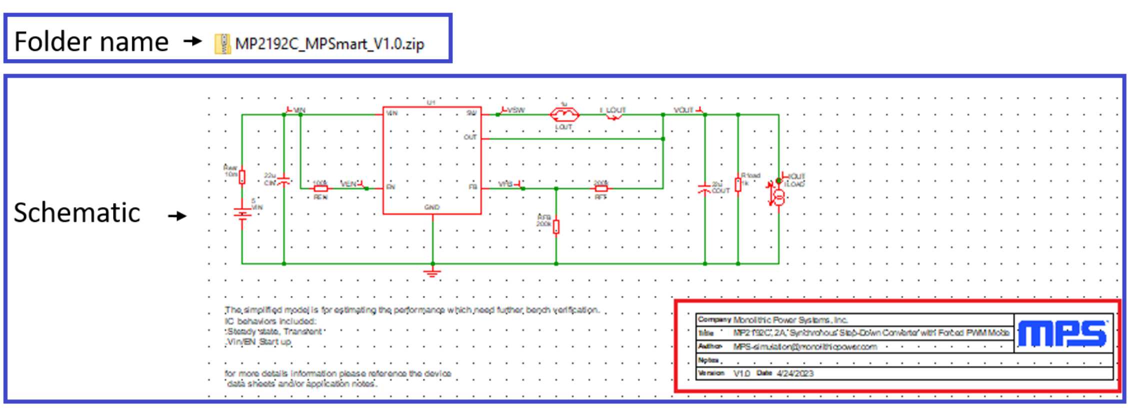


Figure 43: MPSmart model version check

Q11: How can users ensure an MPSmart model's reliability?

A: All models provided by MPS are built by experienced engineers, and are fully compared and verified with experimental results. The models can simplify and speed up the design process. Note that while the models are good for estimating design performance, they are not suitable for further bench verification.

Q12: The simulation result differs significantly from the bench result.

A: One possible reason is that the simulation and bench circuit may not operate under exactly the same conditions. For example, a user may need to consider the derating condition of the capacitors and inductors—check the load transient conditions, such as real load slew-rate on the oscillation.

Q13: Is it possible to use an MPSmart model to calculate efficiency?

A: Yes, the MPSmart models include power loss information, which can be used to estimate the efficiency of a given circuit design.

Note that an MPSmart model can provide useful estimates in predicting the circuit behavior. The type of behavior model may not perfectly consider all real variables and parameters, such as temperature or parasitic parameters, all of which can affect the actual efficiency in real applications. MPS also has other tools for efficiency calculation, such as Spreadsheet and DC/DC Designer (see Figure 44). These tools are available on the MPS website.

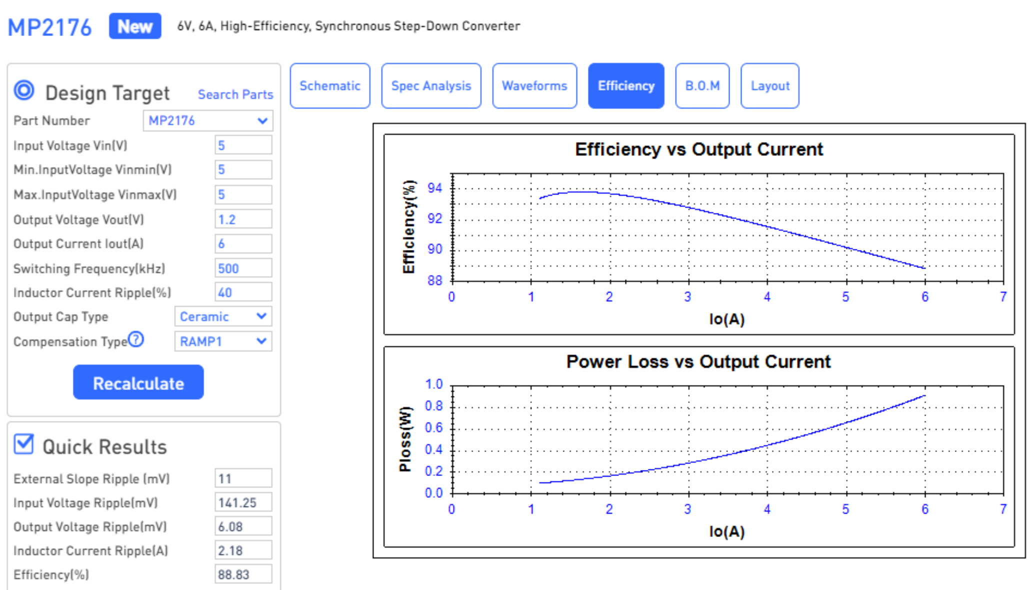


Figure 44: Efficiency Calculation on MPS DC/DC Designer

Q14: The spreadsheet does not work as expected.

A: Most of the spreadsheets we provide require users to enable macros. If the spreadsheet displays "SECURITY WARNING", click the "Enable Content" button (see Figure 45).

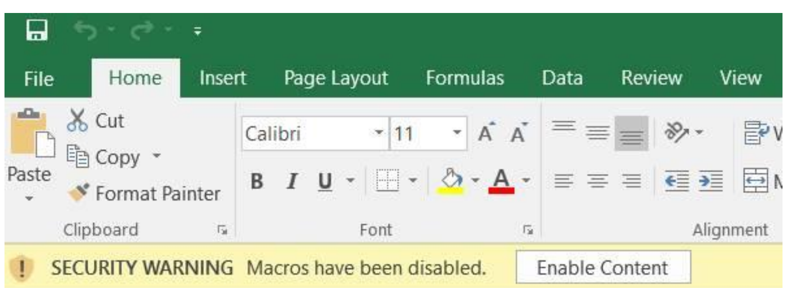


Figure 45: Enable Content for Spreadsheet

Alternatively, check to "Trust access to the VBP project object model" (see Figure 46 on page 33).

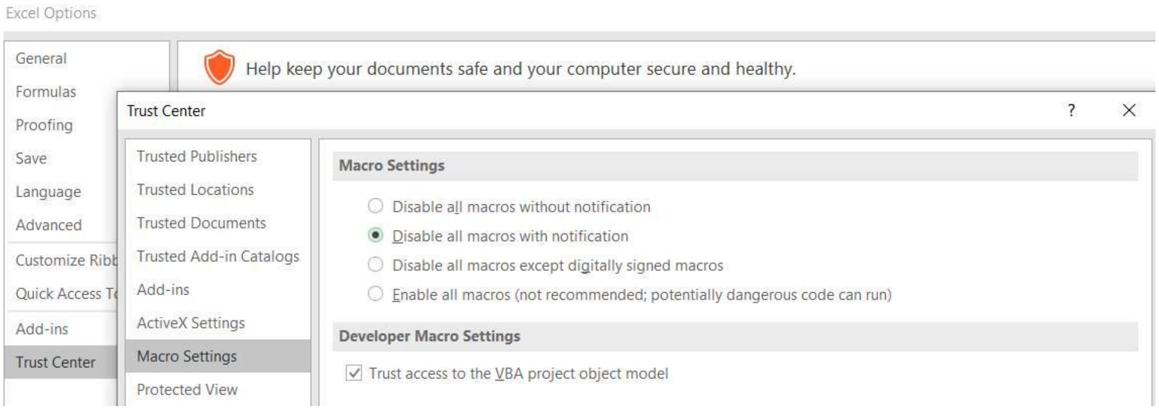


Figure 46: Enable Macro Settings

Contacting Technical Support

If a user is having any problems or questions about using MPSmart that are not covered in this guide, contact the technical support team for assistance. There are several ways to reach out to technical support:

1. Regional FAE: Contact your regional Field Application Engineer (https://www.monolithicpower.com/en/contact.html) for assistance.
2. Online forum: Visit the MPS website's technical forum (https://forum.monolithicpower.com/) and navigate to the support page. Fill out the online form with description of your issue or question. The support team will respond to your inquiry as soon as possible.
3. Email: Send an email to the technical support team (MPS-Simulation@monolithicpower.com). Provide a detailed description of your issue or question, along with any relevant files or screenshots to help the support team understand your concern.

By contacting technical support, you can receive expert assistance in addressing issues or questions about using MPSmart to ensure a smooth and efficient design process.

CONCLUSION

MPSmart is a simulation software that allows users to evaluate the performance of MPS ICs using IC models and simulation applications provided by MPS.

This application note introduces basic features of MPSmart, how to obtain it, and how to install it on a computer. It also describes how to find and download MPS IC models. This application note introduces users to the basic operation and simulation analysis to help get started with MPSmart. It also describes how to use other design tools (spreadsheets and the MPS GUI) to assist in circuit design and speed up the design process. Examples developed for MPS products and typical applications are given to demonstrate the simulation ability of MPSmart. Users can explore the ideas and design concerns with MPS IC products easily and efficiently. The application note also addresses some common issues that users may experience when working with MPSmart and possible solutions.

REFERENCES

1. SIMetrix/SIMPLIS User's manual, version 8.4
2. SIMPLIS Technologies, online material, https://www.simplistechnologies.com/
3. MP6005 datasheet
4. MPM3695-25 datasheet

REVISION HISTORY

Revision #	Revision Date	Description	Pages Updated
1.0	9/14/2023	Initial Release	-