RidleyWorks®

with

RidleyIQ™

USER GUIDE

Manual Version **20.02**
July 7, 2025

Ridley Engineering

For more information, please contact Ridley Engineering directly for assistance. We are here to help you.

E-Mail: info@ridleyengineering.com http://www.ridleyengineering.com

RidleyWorks® Release 20 Toolbar Icons

Release 20 of **RidleyWorks®** has an intuitive toolbar system for navigation around the program. You will see the toolbar above on the **Power Stage** worksheet of the program. If you click on a specific icon, it will take you to a new page, or pull up a form for data entry and design. Each of the toolbar icons are described in this section of the **RidleyWorks®** manual.

The gear wheel icon at the right of the toolbar opens a form to let you choose the design features that you want to use.

This form shows you the standard design features of **RidleyWorks®** and allows you to select optional advanced design features. You can also use the form to open the RidleyBox for measurements.

The voice commands can be turned on and off on this form as you wish.

Specifications	Enter Input and Output Specifications	Topology	Choose Power Converter Topology
Hz	Select Switching Frequency	1..2..3..	Show the Design Sequence
	Show Time-Domain Waveforms	dB	Show Bode Plots
η	Show efficiency details		Design Power Inductor
	Design Power Transformer		Design Two-Stage LC Filter
LLC	Design LLC Power Stage		Design Snubber Circuit
LT	Create LTspice Schematics	Import/Export	Save/Load Design Values
Summary	View Summary Page	Clear	Clear All Design Values

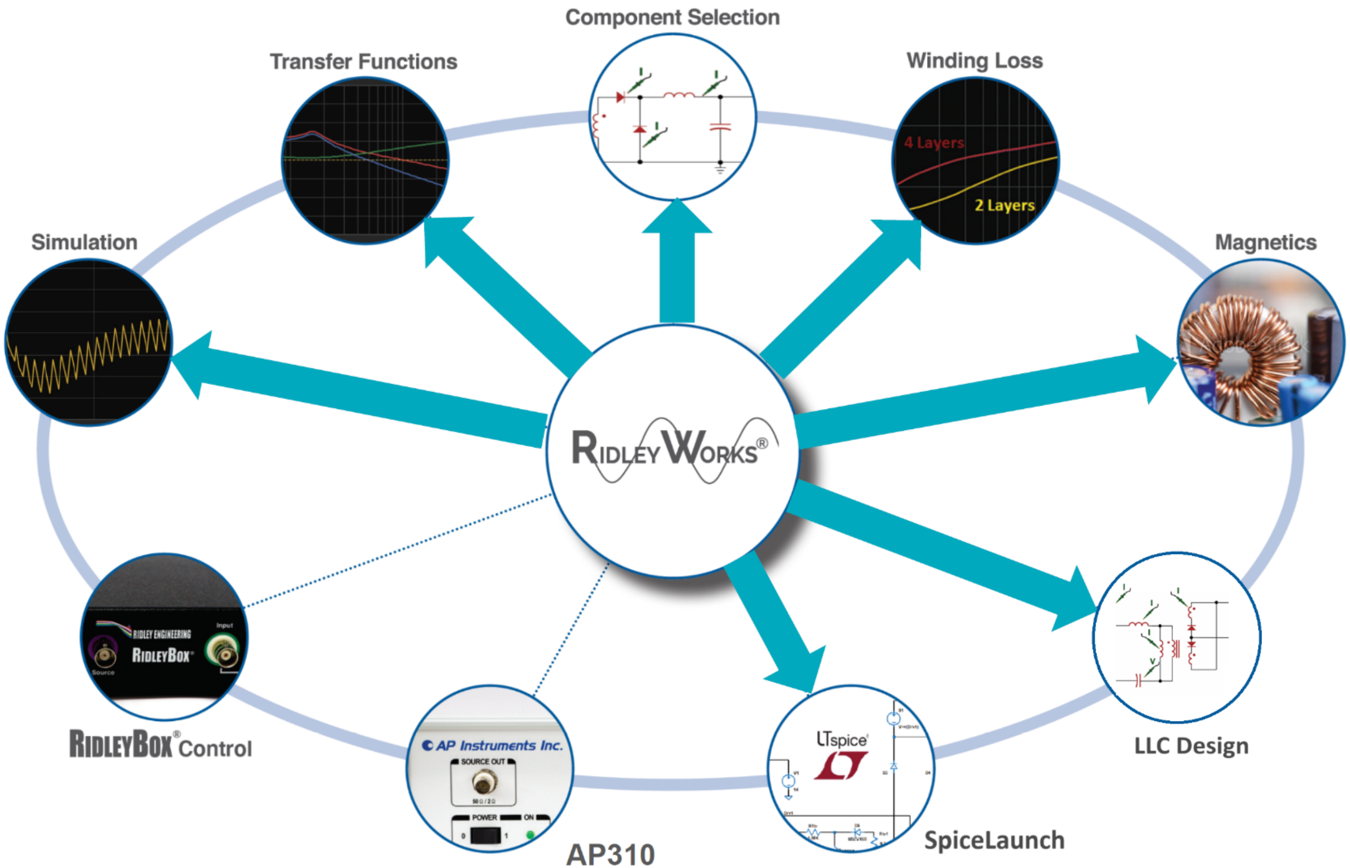
COOL-I-O	Start the RidleyBox Link to collect data
SIMZERO	Simulate converter from Startup
LT Import	Import LLC Simulation Data from LTspice
RIDLEYIQ™	High-level Intelligent Design Feature

Magnetics Toolbar Icons

Schematic	Return to Power Stage Schematic
1..2..3..	Show Recommended Magnetics Design Sequence
	Show Magnetics Impedance Plots
Turns/Core	Select Core and Turns
	Design Windings
µ	Select Core Material
Dowells	Solve for AC Resistance
Clear	Clear Magnetics Parameters Only

Table of Contents

1 Introduction to RidleyWorks®



RidleyWorks® is the only switching power supply design program which provides component design, large-signal simulation, feedback control design, and small signal analysis in one easy-to-use package. Features of **RidleyWorks®**, Release 14 include:

- **Power stage designer**
- **Magnetics designer** (included in full version, not in Control and Simulation version.) Library of cores.
- **36 most popular topologies** used in the industry. All topologies in one program allow you to compare quickly and easily
- Industry's fastest cycle-by-cycle simulation shows true large-signal performance
- **SPICELaunch™** instantly creates **LTspice®** files to continue design and more detailed simulation with all components selected.
- **Unique LTspice® models** for proximity loss and core loss.
- **Control loop designer** suitable for constant-frequency PWM
- **Current-mode control** using the latest and most accurate modeling techniques
- **Voltage-mode control** using the results of the PWM switch model
- **CCM and DCM** converter simulation and analysis
- **Small-signal analysis** of control system, including loop gain
- **Digital compensation** delay calculation
- Two-stage **input filter design** and analysis
- **Output impedance** analysis
- Audiosusceptibility or **PSRR analysis**
- **Second-stage LC** output filter analysis and design
- **Snubber** design and analysis
- **Switching loss** analysis for power FETs and IGBTs
- **LLC power circuit** design for 8 different topologies

There are three levels of application for this software. These levels are commensurate with experience to help you achieve the greatest value from RidleyWorks® without the burden of struggling with the **6500+** design equations involved in arriving at the solution.

Level 1: Basic Design

At this level, you can simply input the power requirements for your circuit. When you click "OK", the entire converter, including control loop, is designed for you. The components are chosen for you based on your data, and these are working values which you can put into hardware with confidence. You can then examine the design with the simulations of RidleyWorks® or convert designs to LTspice® files with our **SPICELaunch®** feature.

Level 2: Intermediate Design

Level two designers have design experience and want to input their knowledge as the design progresses. They want to understand and further optimize their design. At this level, you will look carefully at the selected values from RidleyWorks® and change some of them to improve the design.

Level 3: Advanced Design

Level three designers not only know how to design an effective power supply— they know how to get maximum performance out of their power system for their very specific applications. They have a library of preferred parts that work well for their applications and use these to fully optimize the design.

The magnetics design section works similarly in levels. You can view the transformer or inductor on the main design schematic and

1. Choose from simplified parameters and refer to a magnetics design house for completion.
2. Choose your own cores, wire, and other components and give the completed design to a manufacturer for production; or
3. Design the magnetics from the ground up using your own parameters and knowledge.

Regardless of your design experience, RidleyWorks® allows you to check and cross-check all parameters and results. You can stop at any time during the process and view resulting waveforms and control parameters.

We hope you enjoy the benefits of this interactive design tool. To learn more about the powerful capabilities of the program, we highly recommend attending one of our hands-on Five-Day Power Supply Design Courses.

1.1 Computer System Requirements (PC-Based)

To run RidleyWorks® effectively, any modern computer with at least **Excel 2019 or later** installed will work. The RidleyWorks® code is extremely efficient, and it does not need a high-powered computer. It is also very compact, taking up only about 23 MB of disk space.

Make sure you select the **64-bit version of Excel** when installing.

RidleyWorks® is recommended for use with Windows 10 or later.

1.2 Computer System Requirements (MAC-Based)

Microsoft Office is available to run on Apple computers, but Excel for MAC does not work as it should. You will be able load RidleyWorks®, but the formatting will not show everything properly. To use RidleyWorks® on the MAC, you will need to install a Windows partition and the normal PC-based version of Excel (**64-bit**).

1.3 RidleyWorks® Software Download

The latest version of RidleyWorks® is available for download at

http://ridleyengineering.com/software-ridley/download-RidleyWorks.html

If you are installing for the first time, you should download the full install version, and if you are upgrading, download the update version.

To download the software, you will need a software product key which looks like

RIDLEYWORKS-XXXX-XXXX-XX

1.4 RidleyWorks® Software Installation

Overview

Install **RidleyWorks®** in **C:\RidleyWorks** directory. Permission files will be placed in the directory c:\Users\<username>\Documents\RIDLEYWORKS\CONFIG directory.

Install **LTspice®** in the default directory during installation. The subroutines and symbols needed should be found in the appropriate folder. We recommend using the latest version of LTspice® **24.**

Detailed Procedure

1. Earlier versions of RidleyWorks® should first be removed using the Windows Uninstall utility. This will not remove any of your working files.

2. We recommend deleting the folder C:\ProgramData\AFY\RIDLEYWORKS so that the permission files are updated to the latest version.
3. You should have downloaded a zip file RIDLEYWORKSINSTALL on your computer. Inside this zip file, you will see the installation file:

   RIDLEYWORKSINSTALL.EXE

4. Double click on the RIDLEYWORKSINSTALL application to install in your Documents folder.

4.2 In the target **C:\RidleyWorks** directory, you should see the following files when the installation program is done:
RidleyWorks18xx.xlsm
Transfer.xlsm
RidleyWorks_to_LTspice®

There is also a permission code library directory located at **C:\Users\<username>\Documents\RIDLEYWORKS\CONFIGIQ** contains the license files:

RidleyWorks64IQ.dll
RidleyWorks64IQ.dll.cm
CMServer.exe

The blue .xlsm filename is the main RidleyWorks® program which will run under Excel. The six files shown in green contain the permission code to work with your specific License Key for the program, provided to you when you purchased RidleyWorks®. If you receive an error message that Excel cannot find the .dll files, please refer to section 101 of this manual.

5. To use the SPICELaunch® feature of RidleyWorks®, you will need to install LTspice®. You can download that from Linear Technology's site. Please install LTspice® in the default directory.) Additional LTspice® files, symbols, and subcircuits will be placed in the appropriate folders on your computer when you install RidleyWorks® in step 3 above.

6. Double click on the blue file name to launch RidleyWorks®. When the program begins to load, you may get the message that it contains Macros. Click **Enable Macros.** The first time that you load RidleyWorks®, there will be a registration process. You can make copies of this file as you wish on your computer. It is often useful to keep the RidleyWorks file on the desktop or in a specific project file.

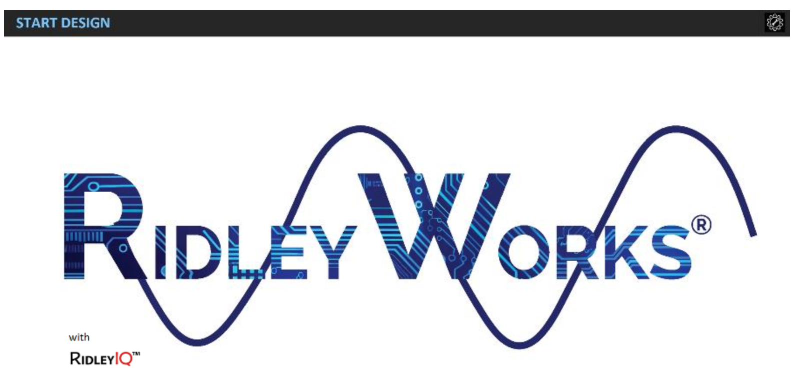
Select the file to be configured as a standalone program and enter the password which you should have received upon purchase. After this, you will be asked to enter registration information, including address and contact information.

(If you receive an error about RidleyWorks.dll files not being found, please refer to page 78 of this manual.)

1.5 RidleyWorks® Software Upgrades

On the opening page of RidleyWorks, you will see the option to Check for Updates. Click on this link to see if there is a later version available for download.

1.6 Loading Workbooks



After registration, Excel will then proceed to open the specified workbook, and a page like that shown in Fig. 1.1 will appear. Once you see this the words "License Verified" in the bottom left corner, your product is fully registered.

To make sure sizing is correct for your display, click on **Screen Size**, and select the option that fit your system. This command is under the gear  icon.

You can also use the zoom slide bar at the bottom right of Excel to adjust individual worksheets.

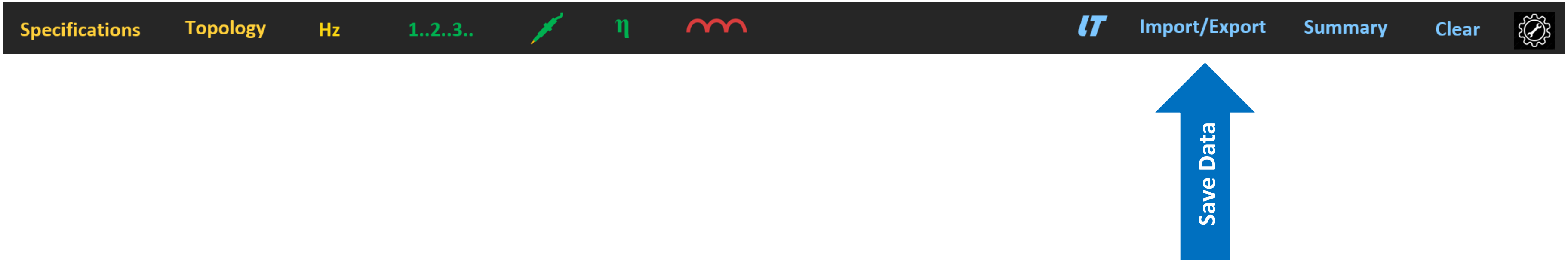
Fig. 1.1: RidleyWorks® Guide Page

1.7 Saving Workbooks

Your design in RidleyWorks® can be saved just like any Excel workbook. It is recommended that you save a project under a different name from the original file. To do this, click on File Save As, and specify a folder and a name for your project.

Every time you do this, you will be saving a complete copy of RidleyWorks® together with the simulator, designer, and all the features of the program. It is a live copy of your design, constantly analyzing and simulating any changes you make, not just the data.

1.8 Saving Data Only and Transferring Data to New Version



It is possible to save just the data from your design in a much smaller file. This feature has been added to allow you to easily transition your design to a new version of the program without having to re-enter all your design choices. You can also send a design to your colleagues in a very small file.

To take advantage of this feature, it is necessary to have two files open – RidleyWorks18.xlsm and TRANSFER.xlsm. (the Transfer file should automatically open.) Once you have opened these files, go to the Power Stage page of RidleyWorks® and click on the Export/Import Data button. When you then click the Import/Export icon, your data that you have entered will be placed in the TRANSFER.xlsm file. Clicking on Import will load data from the TRANSFER.xlsm file into your RidleyWorks® worksheet.

You can then open the latest version of RidleyWorks® and follow the process above to reload your data into the new version.

1.9 Which Version Do I Have?

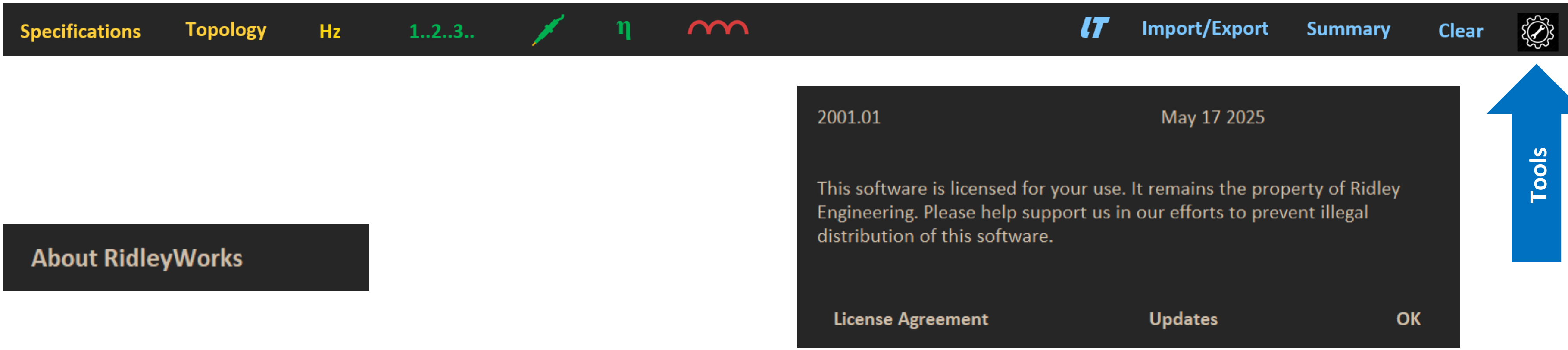


Fig. 1.2: RidleyWorks® Version Number and Recent Updates.

To find out which version you have installed, click on the gear wheel tool on the toolbar, the on About RidleyWorks. You can also see the recent changes and upgrades by clicking on the **Updates** button. The version number will also be displayed on the opening screen.

1.10 Help Notes

RidleyWorks® has extensive on-line help inside the program. Click on any of the help buttons to access notes about a particular feature of RidleyWorks®. A wealth of design information is contained in the program, together with power supply design tips.

1.11 Software Support

If you are having problems installing or running RidleyWorks®, please call or e-mail for help. If no one is immediately available, please leave a message, including your name and phone number, and your call will be returned as soon as possible. You can also choose to attach your RidleyWorks® file to an email for assistance.

Phone: (US) 805 504 2212 E-Mail: info@ridleyengineering.com

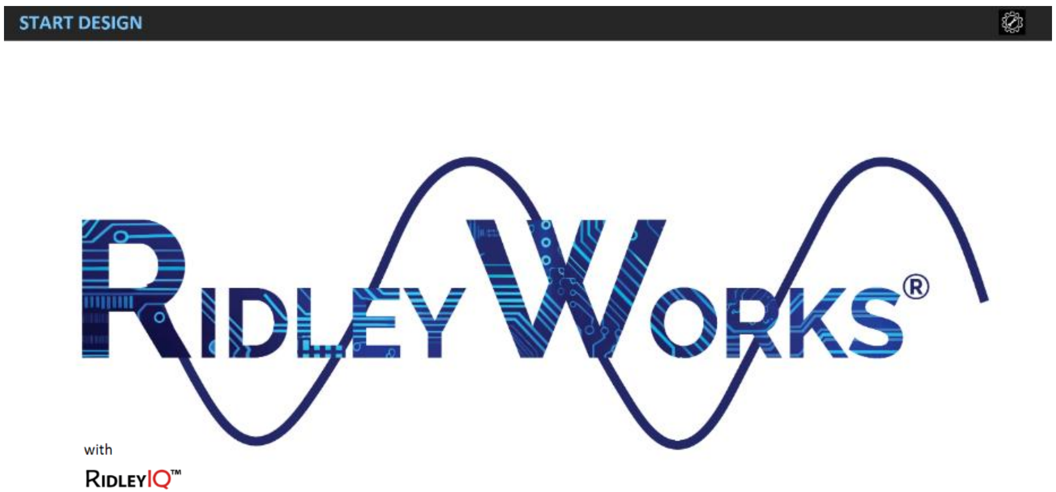
(UK) +44 (0)1509 276 245

1.12 Facebook Support Group

If you are a part of Facebook, there is a group dedicated to power supply design with over 7500 industry and research members. Whenever a new version of the software is available, a posting will be placed on this site, immediately notifying you. You can also ask questions about the software at this site. The group is called

POWER SUPPLY DESIGN CENTER

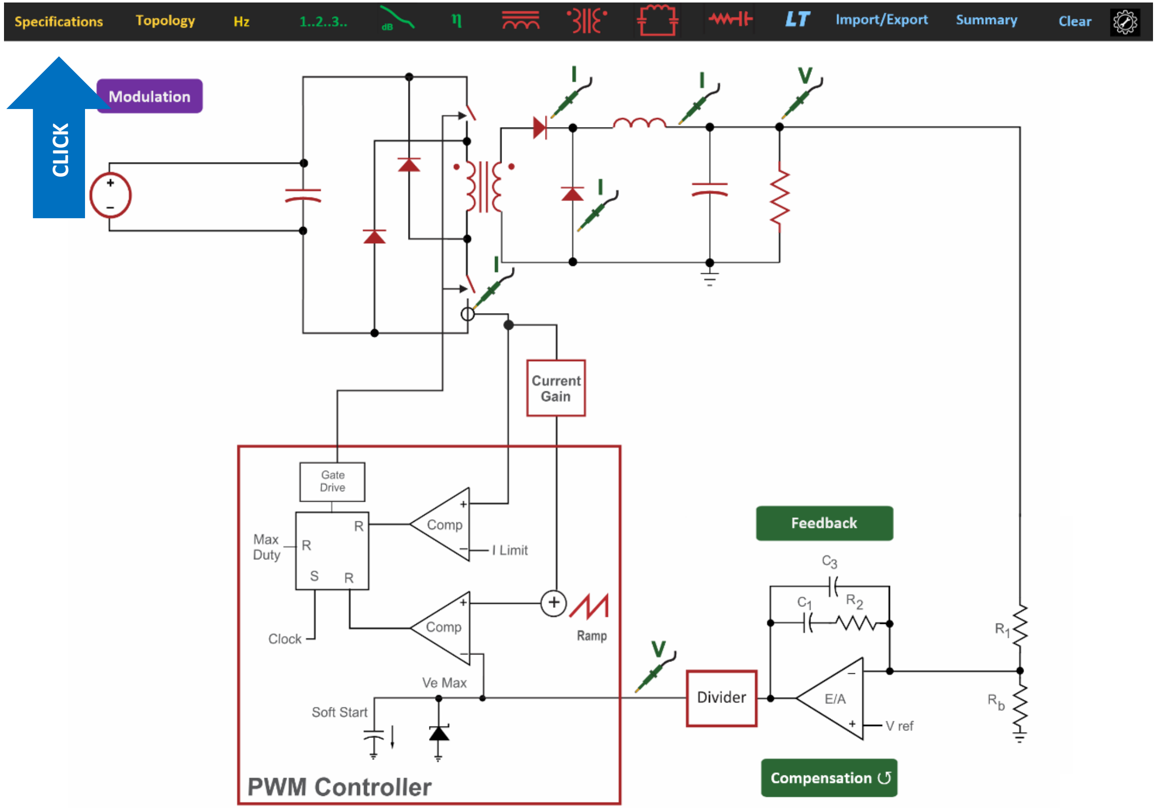
2 Design and Simulation



Click on **Start Design**, and you will be taken to the schematic page to begin your design process

Fig. 2.1: RidleyWorks® Guide Page

2.1 Entering Specifications and Topology



You will start working on the schematics page as shown below. This will let you see waveforms and small-signal measurements while interactively changing power parts and control parts. An automatic expert-system design routine will help you with all component choices. If you are a new designer, you may choose to let RidleyWorks® choose every parameter for you and you can expect a good working design. On the schematic page, you will see a power topology, a controller with all the settings needed to implement most constant-frequency options, and a feedback compensator. The compensator and controller may all be in one chip, and you have full control of all the parameters.

Schematic components in red are clickable, and you can edit their values and see components stresses. This will be demonstrated in more detail later. First, though, you will want to enter your specification for your power system. To enter your power specifications, click on the Spceification Icon as shown.

Fig. 2.2: RidleyWorks® Schematic Page

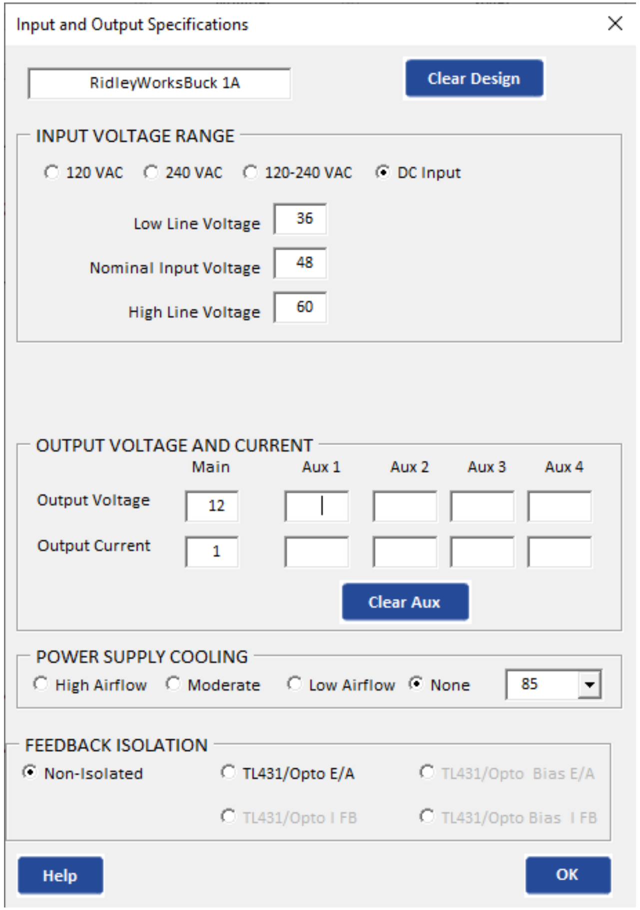


Fig. 2.3: RidleyWorks® Specifications Entry Form

The first area of the Specifications form lets you choose either DC or AC input. You can select the pre-programmed ranges of ac or enter specific voltages that you want to use. One of the unique advantages of RidleyWorks is that there is no limit on the voltage ranges. The design process is not restricted to any specific controller that may limit the operating ranges. Control chip choice, and semiconductor choice, will come later.

If you plan on using a flyback converter, you can specify up to 5 outputs. Other converters are restricted to single outputs in the present release, although this does not limit the experienced designer. You should use coupled inductors to provide the extra outputs. The single output for other converters will be designed at the sum of all the power outputs.

Airflow and temperature ranges will impact magnetics design. The higher the temperature, the larger the magnetics size estimates will become.

The feedback can be configured as either non-isolated, or with a TL431 optocoupler.

Once you click OK on this form you will see a summary of your inputs to check, then you will be presented with a set of choices for the topology of the power stage.

The topology choices will be to the right of the main schematic. You may have to scroll right to see these after clicking on the Topologies icon in the toolbar.

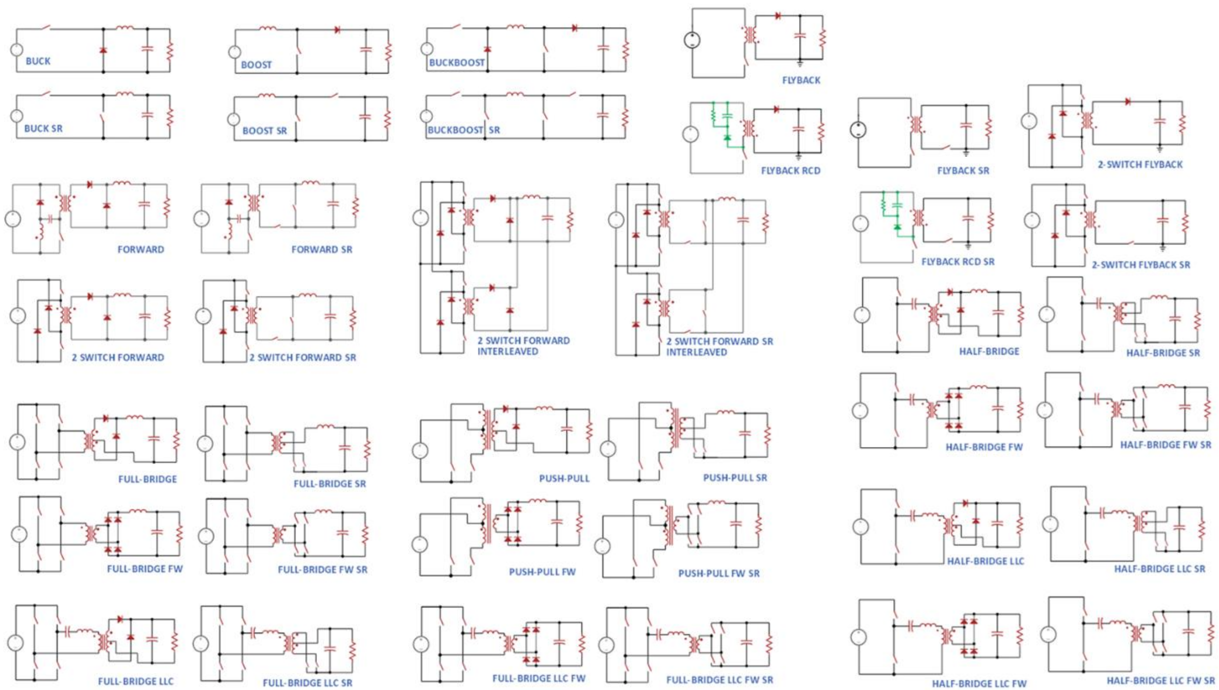


Figure 2.4 shows the full set of topologies available in RidleyWorks®. For ease of use, these are grouped into families for you. Once you have selected a new topology, RidleyWorks® will proceed to do an amazing amount of work. First, all the power components will be selected. A controller will then be optimized around the power stage to give good transient performance and stability. Loop gains will be plotted and analyzed. Finally, 800 cycles of large-signal operation will be simulated. All of this is done instantly. This will totally change your design and simulation experience. Your design procedure will become much more interactive, trying far more options and repetitively simulating as if you had a live breadboard.

Before digging deeper into design values, it is very valuable to look at circuit waveforms to see how the converter is operating. At this point, you have had no input to design values and RidleyWorks® will provide you with fully automated results for a realistic converter. You can verify this by looking at the waveforms of the circuit.

Fig. 2.4: RidleyWorks® Topology Choices

2.2 Running Waveform Simulations

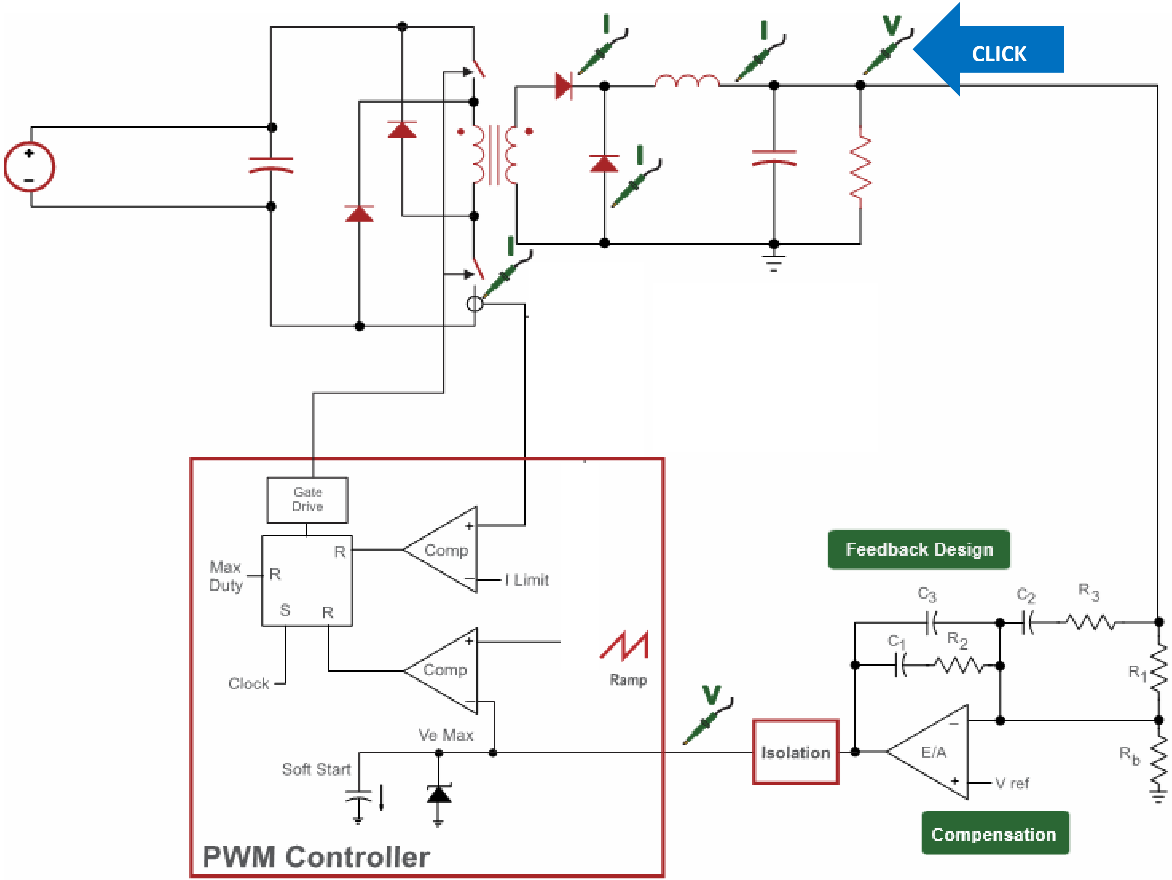


Fig. 2.5: RidleyWorks® Waveform Choices

Figure 2.5 shows the selection of voltages and currents that can be probed with RidleyWorks®. The simulation algorithm inside RidleyWorks® is so fast that you will routinely run hundreds of simulations during a design session, never having to wait for the results. Every time a component is changed, another 800-cycle simulation is run to confirm that the converter is operating properly.

To begin doing this click on the blue button labeled Waveforms, then click on the scope probe at the output of the converter.

Fig. 2.6: RidleyWorks® Output Voltage Waveform

RidleyWorks® automatically simulates 800 cycles at a time. To zoom in on this waveform, treat the controls of RidleyWorks® just like a scope. Turn the **Time Base** dial clockwise to zoom in by clicking on the up arrow next to the dial. Each time you click on the up arrow, the number of complete waveforms will be cut in half. You can quickly turn the dial back to the original setting by clicking on the Min setting.

2.3 Startup Simulation

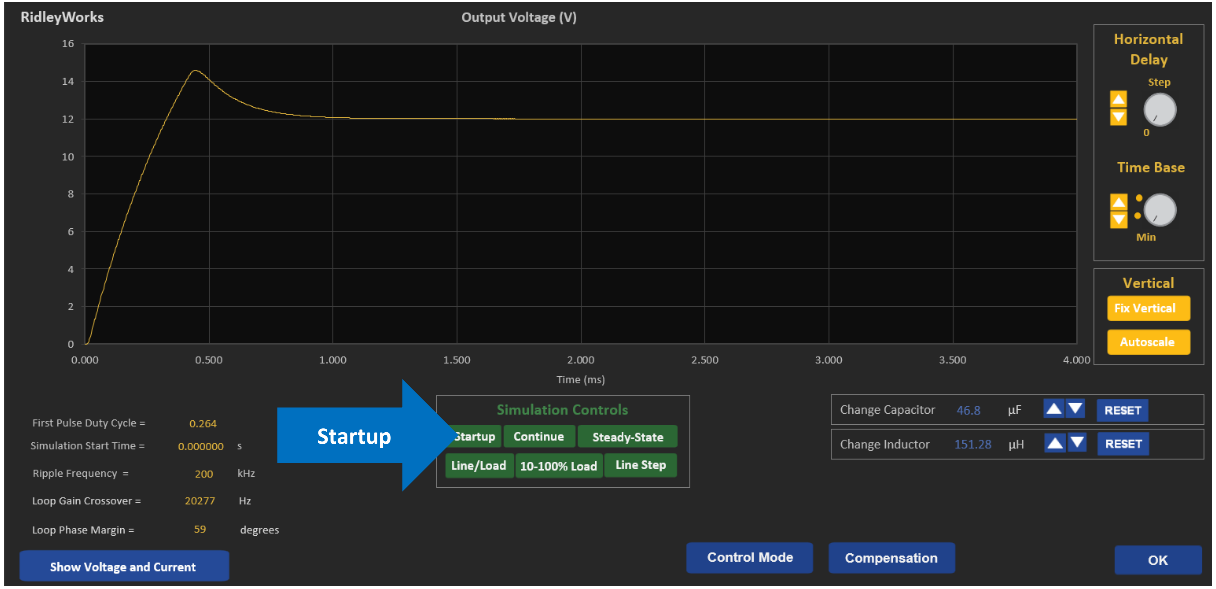


Fig. 2.8: RidleyWorks® Start Up Simulation Voltage Waveform

Startup of your converter can be simulated by clicking the green button as shown above. This will start with zero output voltage, and zero inductor current, and simulate the converter into regulation as shown. It takes over 200 cycles to get the output voltage into regulation in the example shown. It can be instructive to click on button to add the current waveform of the converter to show what effect the current limiting has on the performance of the startup.

Fig. 2.9: RidleyWorks® Start Up Simulation Voltage and Current Waveforms

In the red waveform of Figure 2.9, you can see that the converter enters current limiting during startup.

Determining the proper values of parameters such as current limit is crucial at the very early stages of design. The peak current will impact the design of magnetics and sense resistors, and you don't want to discover this kind of effect later in the design cycle. RidleyWorks® is ideal for this kind of work since it automatically designs all compensation parameters at the very beginning of a design, and this is essential for proper simulation.

2.4 Step-Load Transient Simulation

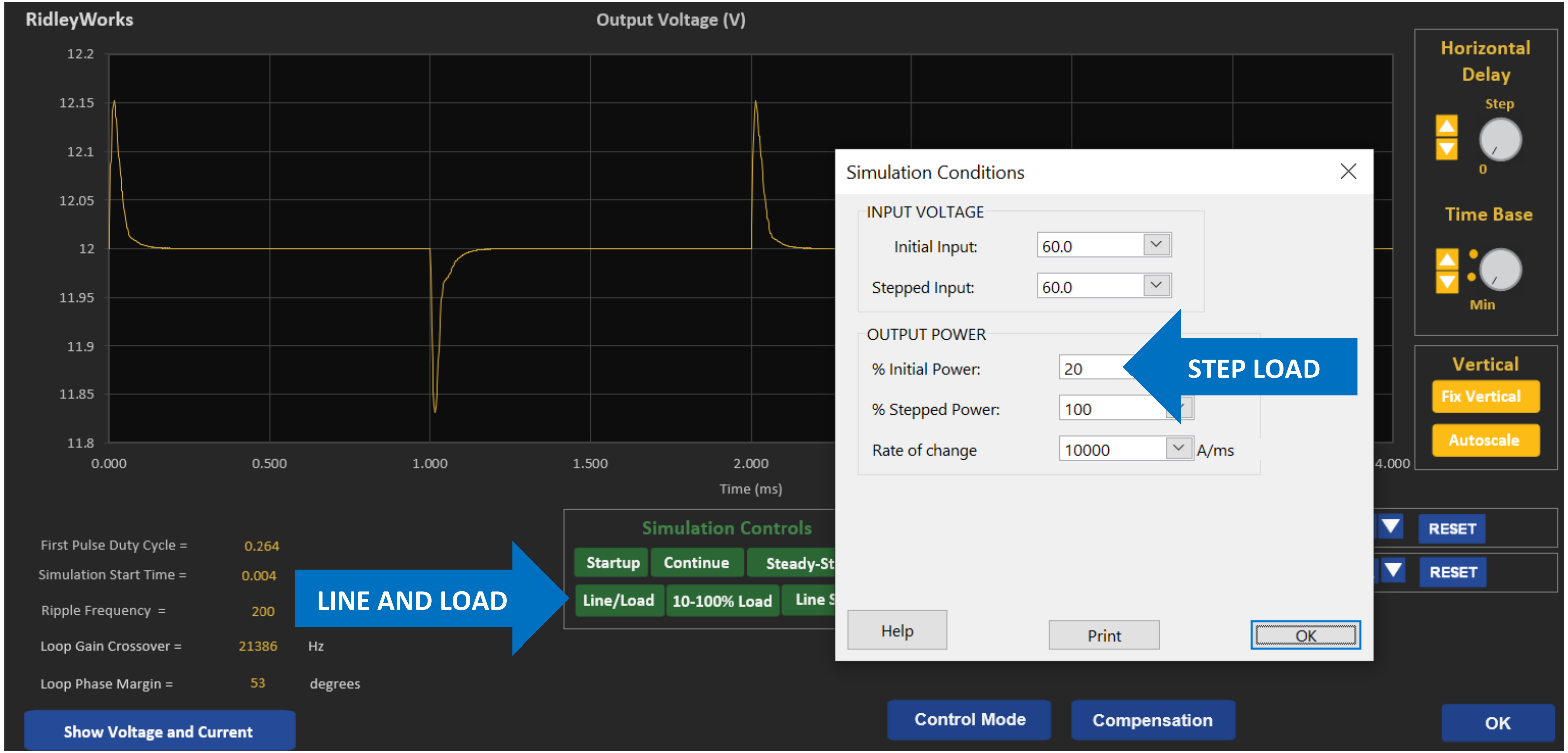


Fig. 2.11: RidleyWorks® Step-Load Transient Simulation

The line and load conditions can be set as shown. Two values of load are shown. The first 200 cycles are run with 100% load, the second 200 cycles with 20% load. Simulation of both the voltage and current are shown, with the inductor current entering DCM on the second transient when the load falls to 20%.

2.5 Adding Input Voltage Modulation

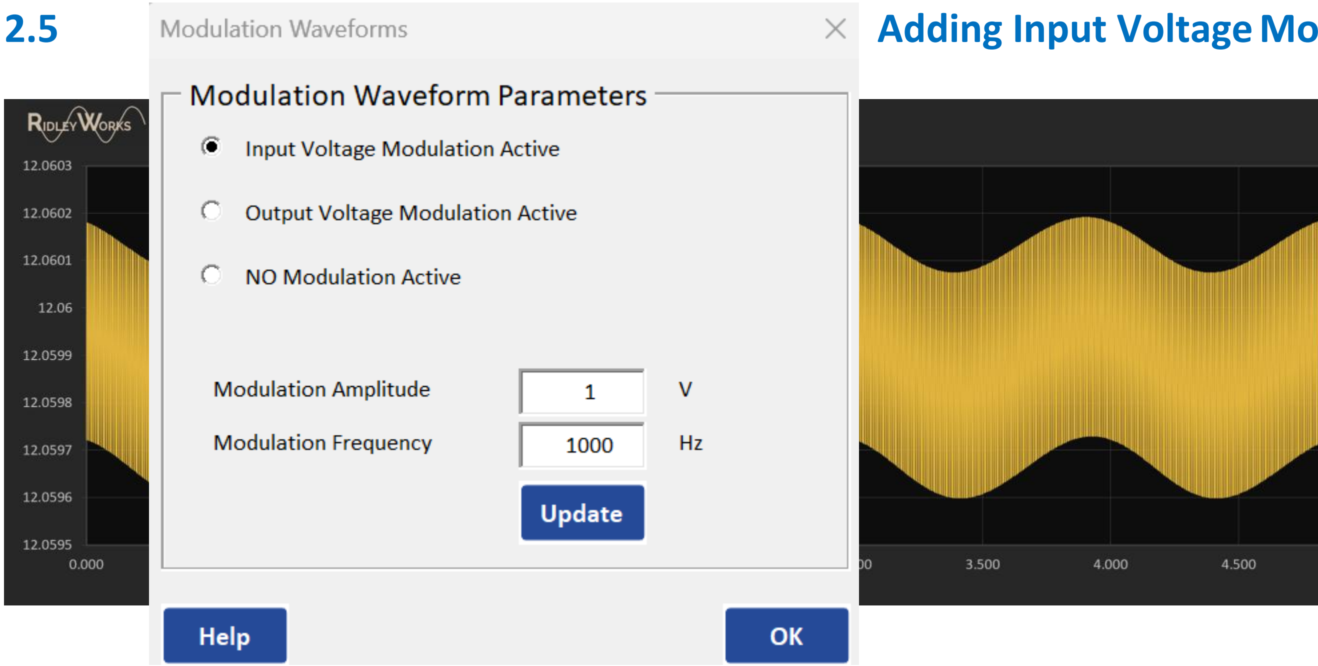


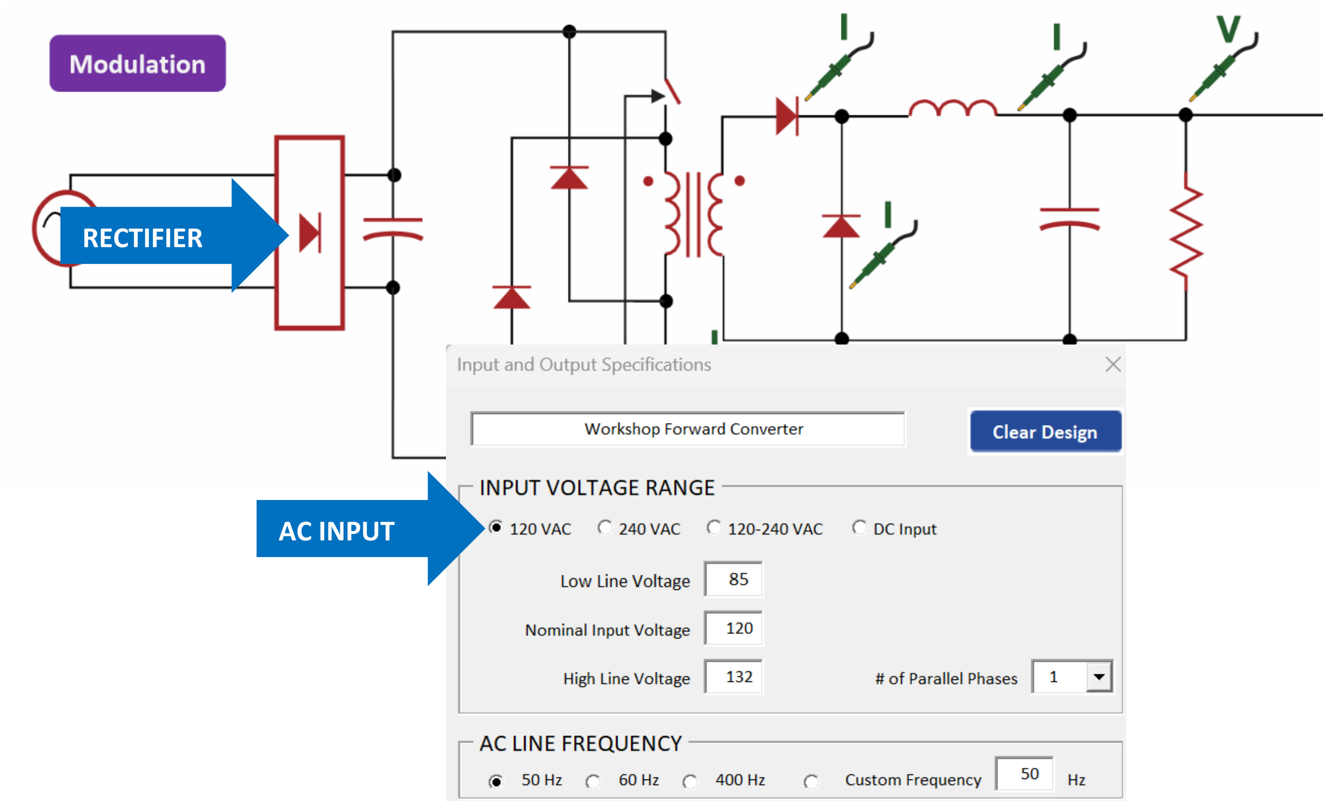
Fig. 2.12: RidleyWorks® Input Line Modulation

RidleyWorks® has the capability to add sinusoidal modulation on top of the input dc voltage. It can also be used to add modulation to the reference voltage to drive the output.

Click on the Modulation button on the schematic to activate this feature and adjust the modulation amplitude and frequency.

The simulation algorithm built into RidleyWorks is fast enough to be able to see the large-signal modulation with the closed loop converter.

2.6 AC Input Simulation



With RidleyWorks® you can select an AC input and perform a simulation of the input rectifier. Three choices of range are available to you. When you check one of these buttons, as shown below, the expected low, nominal, and high-line voltages will be filled in for you. You can change these values if you wish. Once you have checked an AC input, the schematic will change on the input, showing you the input rectifier block.

After clicking OK, click on the rectifier at the input to see the simulated input waveforms.

Fig. 2.13: Choosing AC Input on the Specifications Form

The simulation of input line current and input capacitor voltage are shown in the figure above. This helps you size the input capacitor appropriately for the desired amount of ripple at the input of your converter.

Fig. 2.14: AC Input Waveforms for a 27 W Flyback Converter

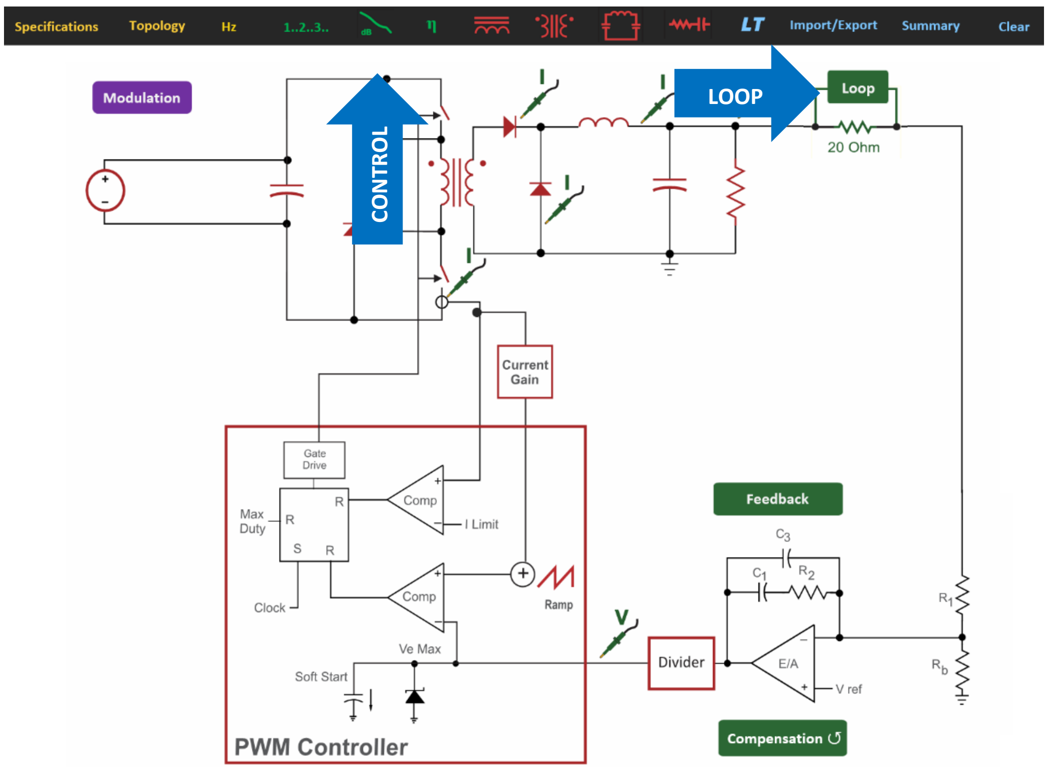
2.7 Exploring Further Simulation Options

Many more simulation options are possible within RidleyWorks® and you are encouraged to try them. For example, with the transient load simulation in Fig. 2.11, you can interactively see the change in transient performance with different value inductors and capacitors. You can move these values up and down while watching the change in the waveforms. This is a truly unique feature of RidleyWorks® that will change the way that you do your designs and greatly speed up the process.

You are encouraged to try all these options in RidleyWorks®. Let us know what you see and tell us if there are additional features that you would like to see included in the program in the future. Once you have a license for the software, all upgrades are free if your license is current.

Please contact us to let us know what you would like to see included in the program in the future. We are constantly upgrading the software to respond to user's needs.

3 Control Analysis Transfer Functions



RidleyWorks® has unsurpassed techniques for designing and analyzing the control system for a switching power supply. The analysis techniques and design routines have evolved over a period of 35 years to give you all the control information that you need to optimize the performance of your power supply.

3.1 Turning On Control Options

When you click on the **Control Design** icon you will see the green options buttons below appear on the power stage schematic. (If you don't see the **Z Out** and the **Audiosusceptibility** buttons, make sure these features are checked when you click on the Features button.)

Once you see the green control buttons, click on the **Loop** button to see the main control transfer functions.

Fig. 3.1: Control Design Options in RidleyWorks®

3.2 Current-Mode and Voltage-Mode Control

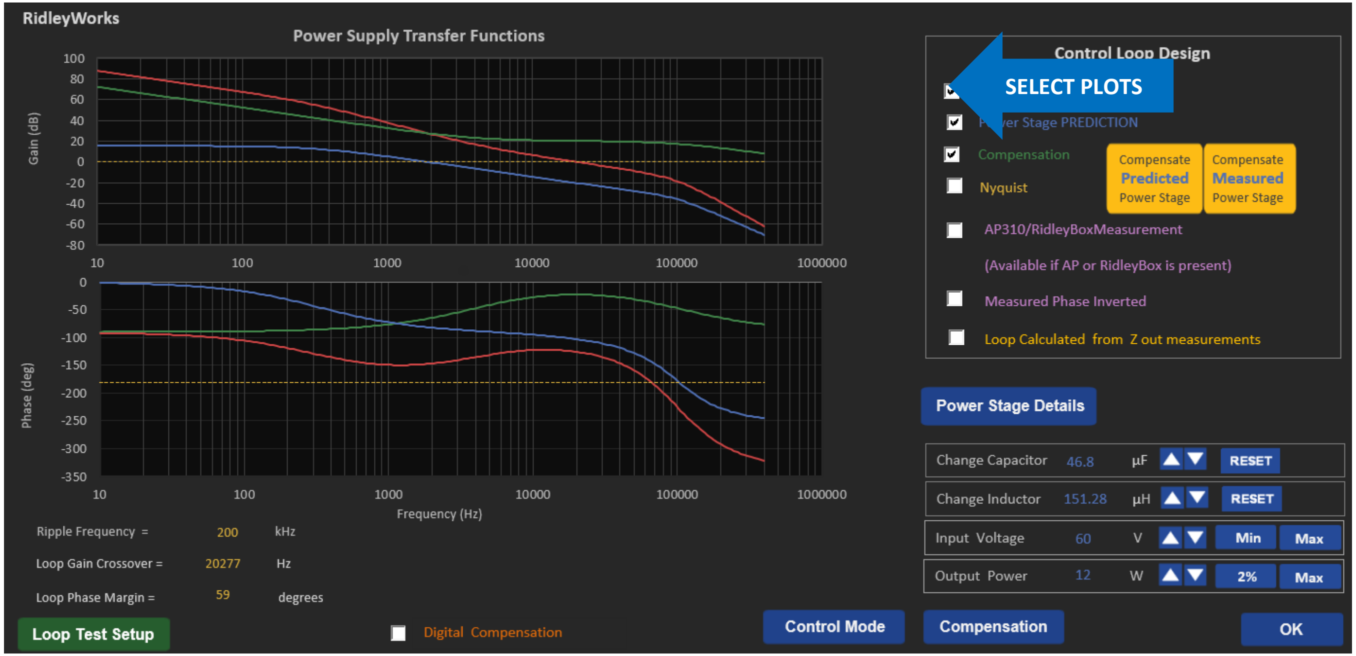


Figure 3.2 shows the control transfer functions which are available in RidleyWorks®. You can select multiple plots to be shown all at the same time as shown in the box above. Measurements from the AP300 or AP310 Frequency Response Analyzer can also be overlaid on the graph to compare with predictions.

You can do many things on these graphs, including moving the line and load up and down, and changing the values of the inductor and capacitor in the converter. As you change these values, you can see the immediate effect on the power stage, compensation and loop gain.

Fig. 3.2: Current-Mode Control Transfer Function Bode Plots in RidleyWorks®

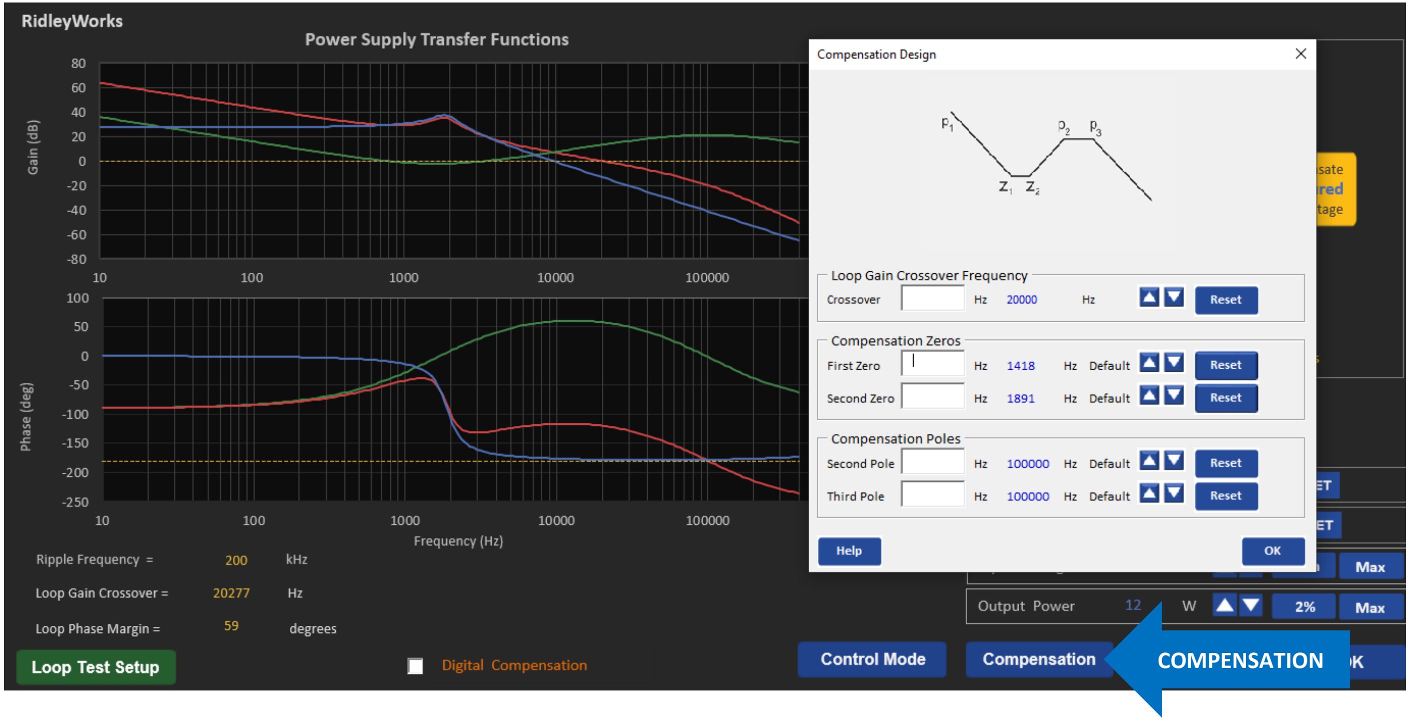
Note that when you change the inductor and capacitor values, each time you click on the button, the control loop is optimized for the new power stage and the transfer function plotted for the updated design. RidleyWorks® will automatically detect whether the converter is operating in CCM or DCM and provide the appropriate transfer function. If you wish to see details of the converter you can click on the **Power Stage Details** button, and it will give you the poles and zeros.

The **Compensation** button is a feature of RidleyWorks® that lets you interactively move the shape of the compensation values while looking at the shape of the loop at the same time. Figure 3.2 shows the transfer functions for current-mode control, the default control scheme chosen for all converters except the half-bridge.

If you click on the Control Mode button, you can change your selection to voltage-mode control. Once you do this, the compensator will be re-optimized and the resulting transfer functions plotted.

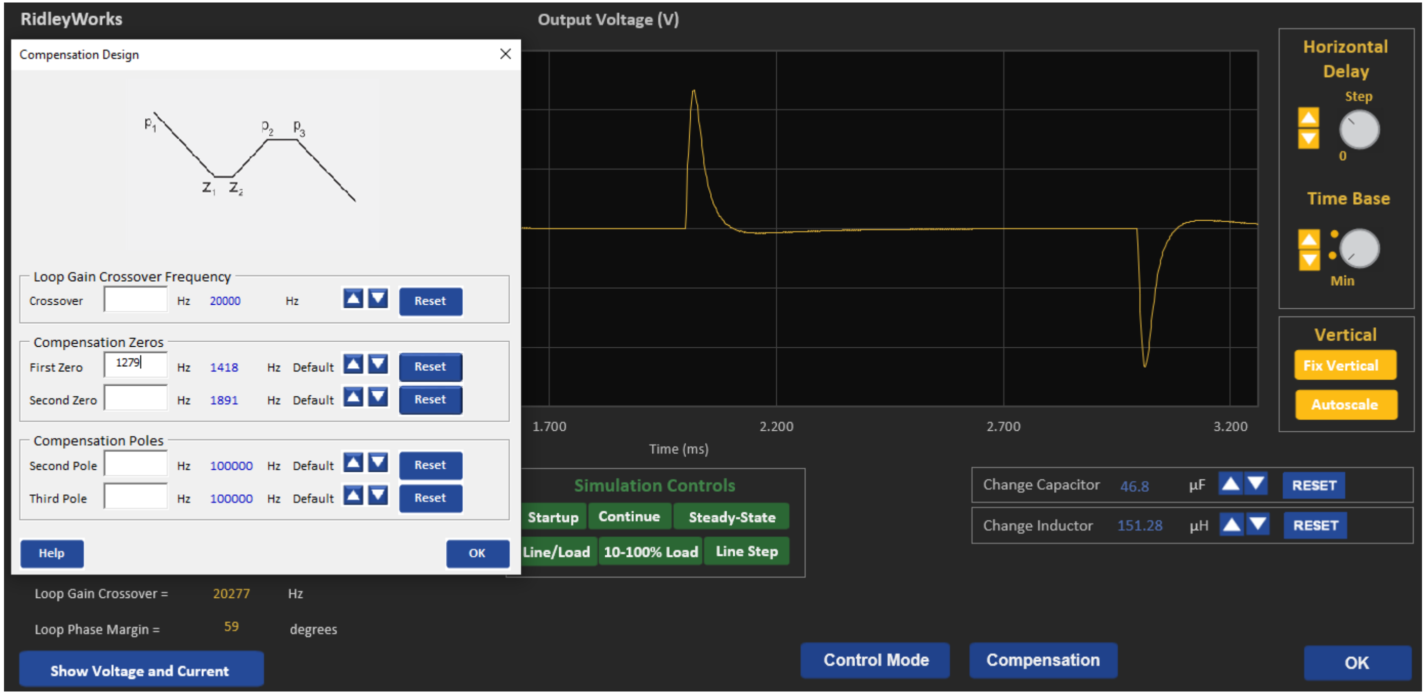
Fig. 3.3: Voltage-Mode Control Transfer Function Bode Plots in RidleyWorks®

3.3 Interactive Compensation Adjustment



It is possible to look at the control transfer functions of RidleyWorks® while adjusting the pole and zero locations of the compensation, as shown in the figure above. Each time you click the **Adjust** button the zero or pole will be moved, new compensation components selected, and the curves replotted.

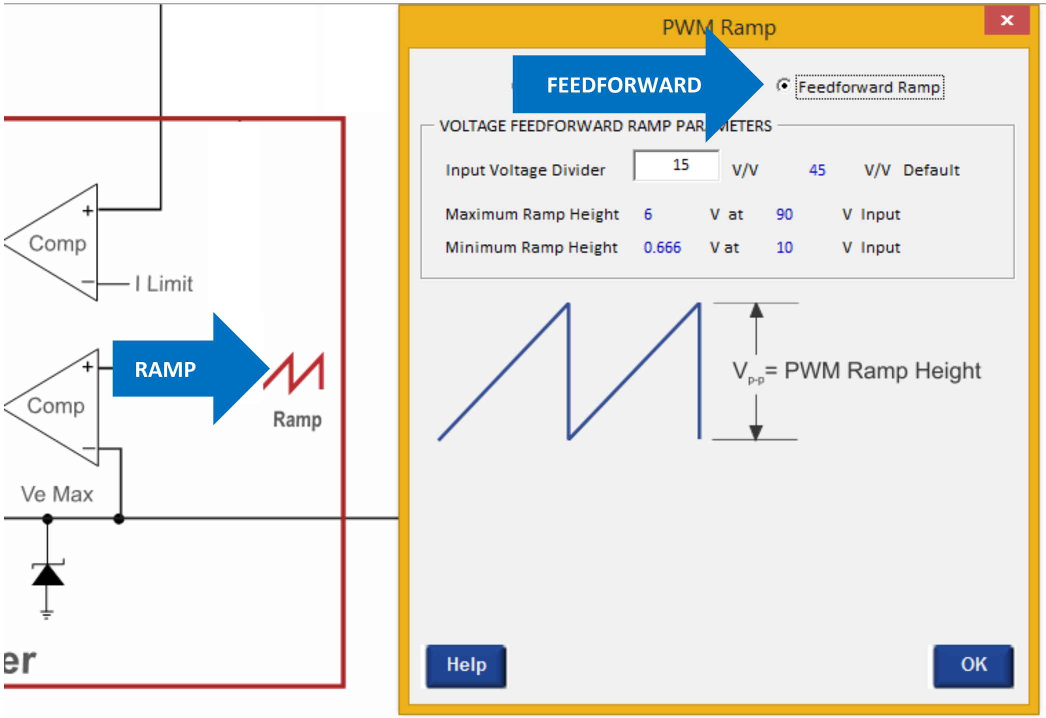
Fig. 3.4: Interactive Adjustment of Compensation Poles and Zeros with Transfer Functions



You can also interactively change the compensation parameters while looking at the simulation waveforms. This is a powerful and completely unique feature of RidleyWorks®. No other design or simulation program has this capability.

Fig. 3.5: Interactive Adjustment of Compensation Poles and Zeros with Simulation Waveforms

3.4 Voltage Feedforward Control



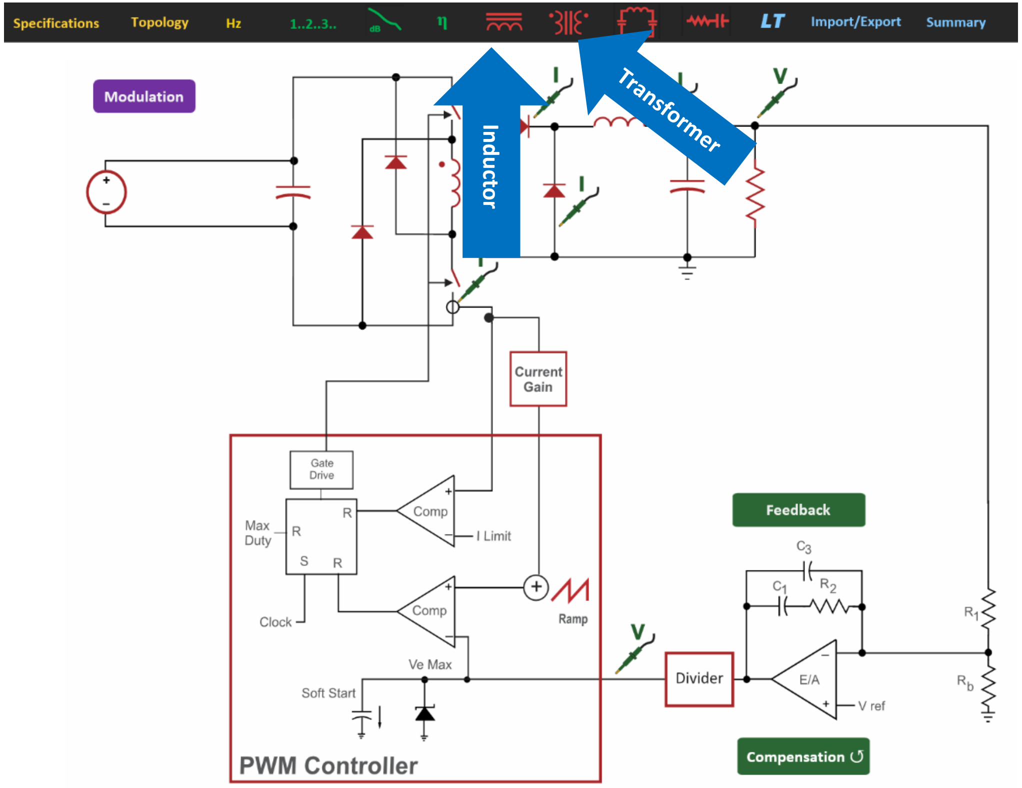
It is possible to implement voltage-mode control with feedforward. The normal implementation of this technique in the control chip is to make the control ramp proportional to the input voltage, scaled by an input voltage divider constant, *k*.

When the input voltage is increased, the ramp size will increase, and there will be an immediate change in duty cycle. This can greatly improve the transient response with changes in input voltage. Normally it will be at least an order of magnitude better, especially for buck-derived converters.

In addition, the changing ramp will make sure that the loop gain of the system stays fixed for different input voltages. This can be very important for large ranges of input voltage, and compensation will remain optimal across the entire range. The converter will regulate much better, and step load response will be improved at low line.

Fig. 3.5: Selecting Voltage-Mode with Feedforward

4 Magnetics Designer



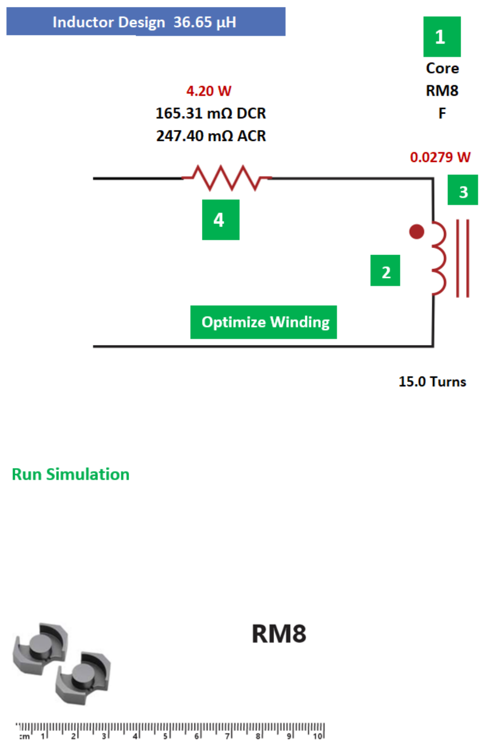
4.1 Turning on the Magnetics Designer

On the Power Stage design page, clicking on the Gearwheel icon button allows you to enable the magnetics design features of RidleyWorks®. You will then see the Inductor Design and Transformer Design icons appear on the toolbar.

Click on either of these to start your magnetics design process.

Fig. 4.1: Selecting the Magnetics Design Option Enables the Inductor and Transformer Designers

4.2 Inductor Design Page



Clicking the Inductor Design icon on the Power Stage page will take you to the inductor design sheet in RidleyWorks®. The detailed inductor design process will include core selection, material selection, winding design, proximity loss and many other details.

If you click on the design sequence icon, you will be guided through the design process with green numbers on the schematic for the inductor.

A crucial part of magnetics development is the need for the frequency response of the component that has been designed. There are links to both the RidleyBox® and the AP310 analyzer from this page.

If you click on Summary, you will see the design table for the inductor. You can provide this to your magnetics manufacturer.

Present Simulation Performance		
Input Voltage	36.00	V
Output Load	60.00	W
Core Loss	0.028	W
RMS Current	5.037	A
AC Current	0.349	A
DC Current	5.025	A
Winding Dissipation	0.34	W
Winding Surface Area	4.11	sq. cm
Total Loss	0.37	W
Inductor Efficiency	99.4	%

Inductor Design		
Inductance	36.65	μH
Peak Current Limit	6.38	A
Core Type	RM8	
Core Area	0.54	sq. cm
Core Material	F	
B max	0.29	T
Approximate Gap	0.41 / 16.4	mm/mils
AL Value	162.86	nH/n2
Inductor Turns	15	
Wire Type	Magnet Wire	
Wire Size	18	awg
Layers	2	
Strands	1	
Each End Margin	0.00	mm
Insulation	0.000	mm
Window Used	66.2	%
DC Resistance	12.43	mOhm
AC Resistance	223.56	mOhm

Fig. 4.2: Inductor Design Page Guides You Through Inductor Details

4.3 Choosing the Inductor Core Size

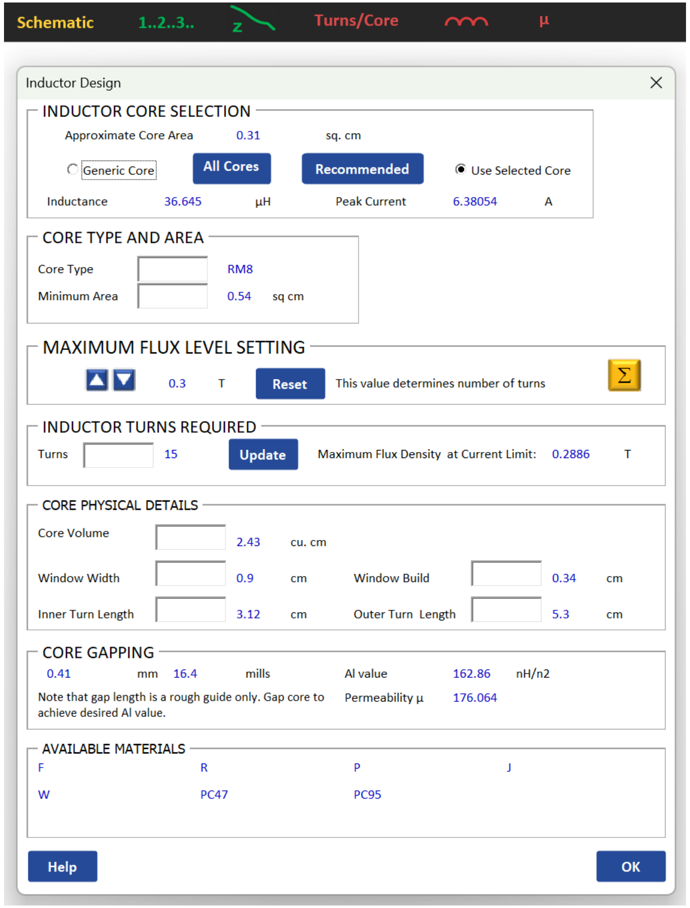


Fig. 4.3: Inductor Core and Turns Selection

Clicking the Turns/Core icon on the inductor design page will bring up the form for selecting the design parameters of the inductor. By default, a generic core will be selected with a recommended core area. This lets you choose any specific core that you or your company likes to use that has approximately that area. A larger core area will run a little cooler, and a smaller core size will have higher dissipation. Your actual selection will depend on your cooling situation, and your need for efficiency versus size.

If you click on the Select Core button, you will be presented with an array of standard cores to use, and you can also add your own favorite cores very easily to the program.

4.4 Choosing the Inductor Core Material

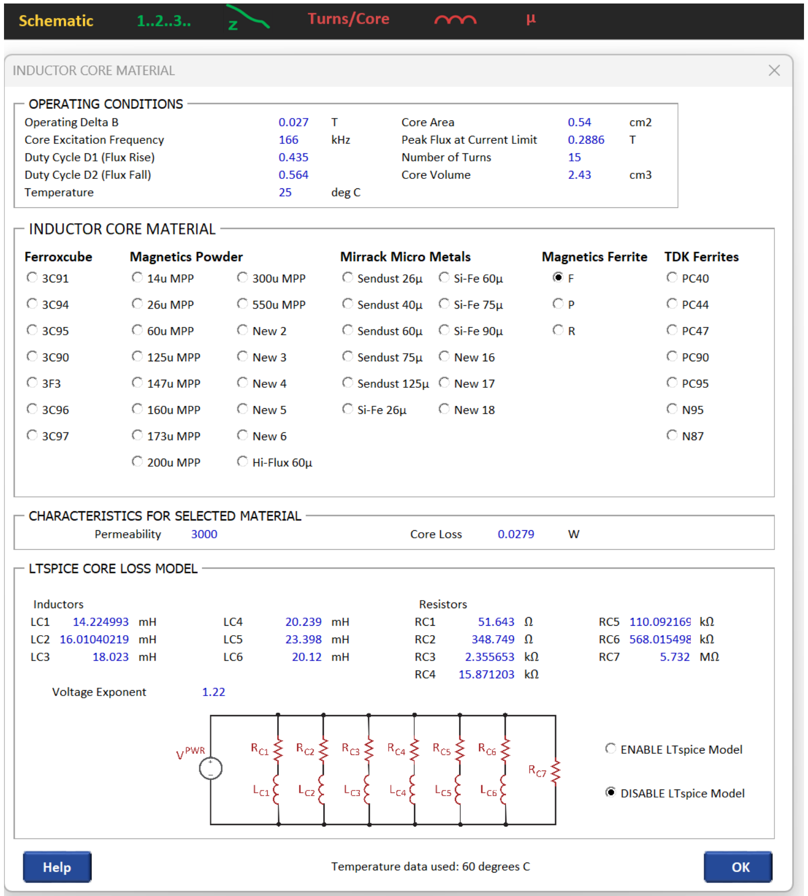


Fig. 4.4: Inductor Core Material Selection

You can use any core selection with a variety of core materials. RidleyWorks® incorporates a unique core loss modeling techniques that accurately give the loss for different materials. Variable Steinmetz equation coefficients are used to produce a much more accurate core loss estimate than is available from the manufacturers. Details of the techniques used to calculate the loss are given in the Ridley Engineering Design Center (www. http://www.ridleyengineering.com/design-center.html) in paper **[A03] Modeling Ferrite Core Losses**.

If you are using a different core material to those shown, RidleyWorks® gives you the proper frequency, flux level, and Delta B to be used with core loss curves.

RidleyWorks® also selects the values of an RL network to model the losses in LTspice®. This is a nonlinear frequency-dependent loss model that allows you to run an LTspice® simulate and click on the core to see the losses. There is no need to use core loss curves.

4.5 Designing Inductor Windings

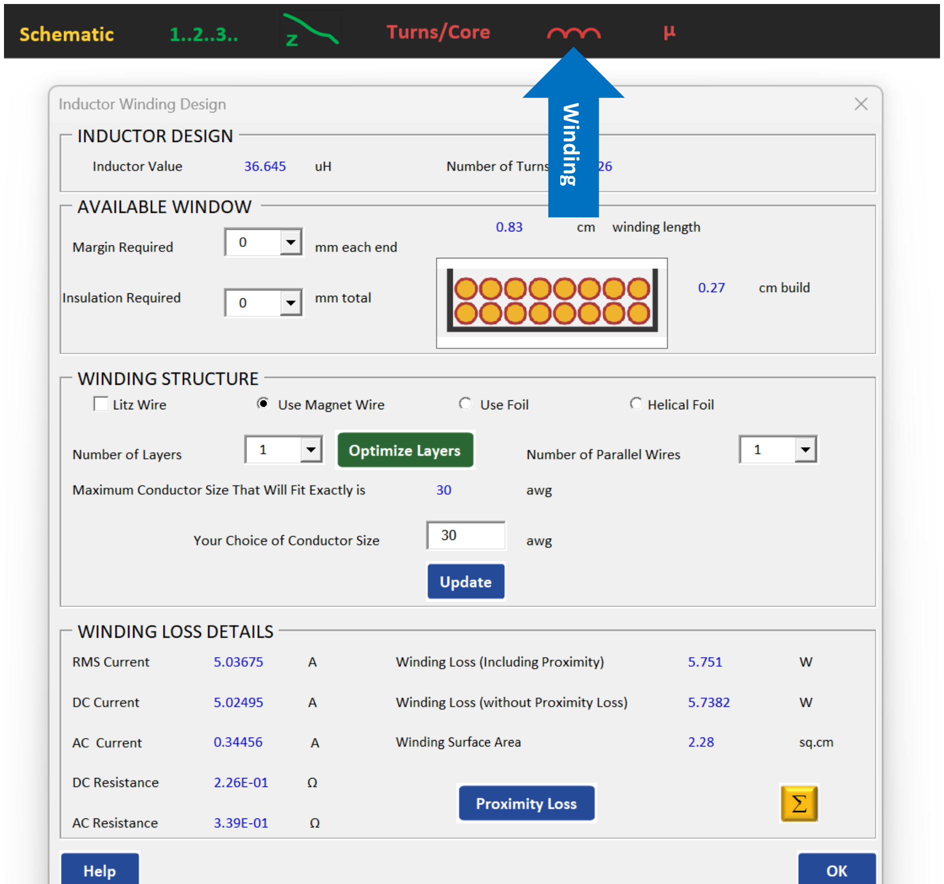


Fig. 4.5: Inductor Winding Design

Pushing the Winding button will show you the details of the inductor winging. You can build inductors with foil, helical windings, or with wire, and RidleyWorks® will help you to optimize each of these. The foil winding thickness can be optimized with proper consideration for proximity losses, by clicking the Minimize Loss button.

Details of the sophisticated techniques used to calculate proximity loss are given in the Ridley Engineering Design Center (www. http://www.ridleyengineering.com/design-center.html) in numerous papers on magnetics design and analysis.

Each of these steps have now been fully automated with RidleyIQ™. If you are not comfortable with the manual design process, just click on the icon to see the RidleyIQ™ features.

4.6 Designing Transformers

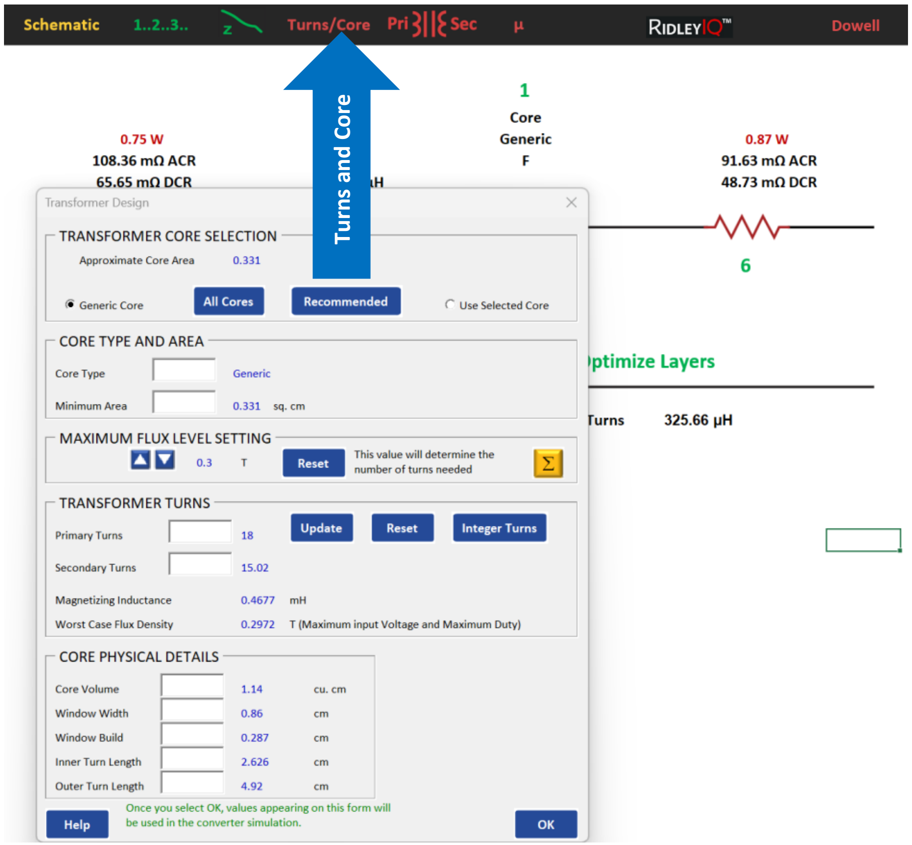


Fig. 4.6: Transformer Design Page

RidleyWorks® has a powerful algorithm for designing transformers, incorporating advanced core loss calculation, proximity loss analysis, core and winding selection, all in an easy-to-use package. The design program interfaces closely with simulation waveforms to provide fast, efficient, and reliable designs.

The sequence of design is similar to that of the inductor. Start by clicking the Turns and Core icon to begin the design.

Each of these steps have now been fully automated with RidleyIQ™. If you are not comfortable with the manual design process, just click on the icon to see the RidleyIQ™ features.

5 SPICELaunch™

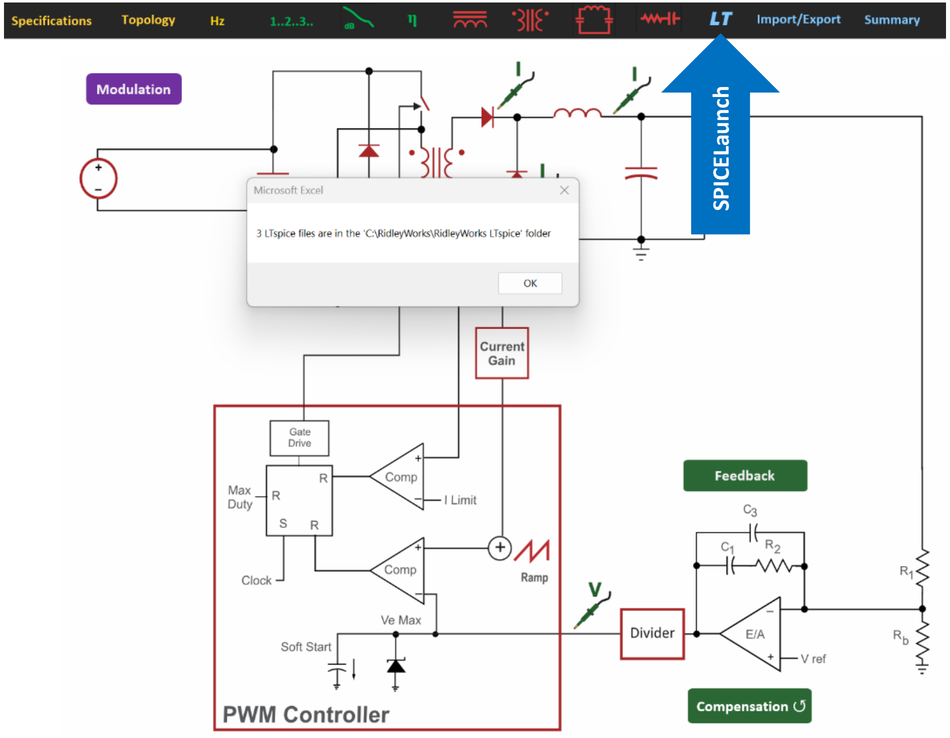


Fig. 5.1: Running SPICELaunch™

5.1 Preparing an LTspice® File

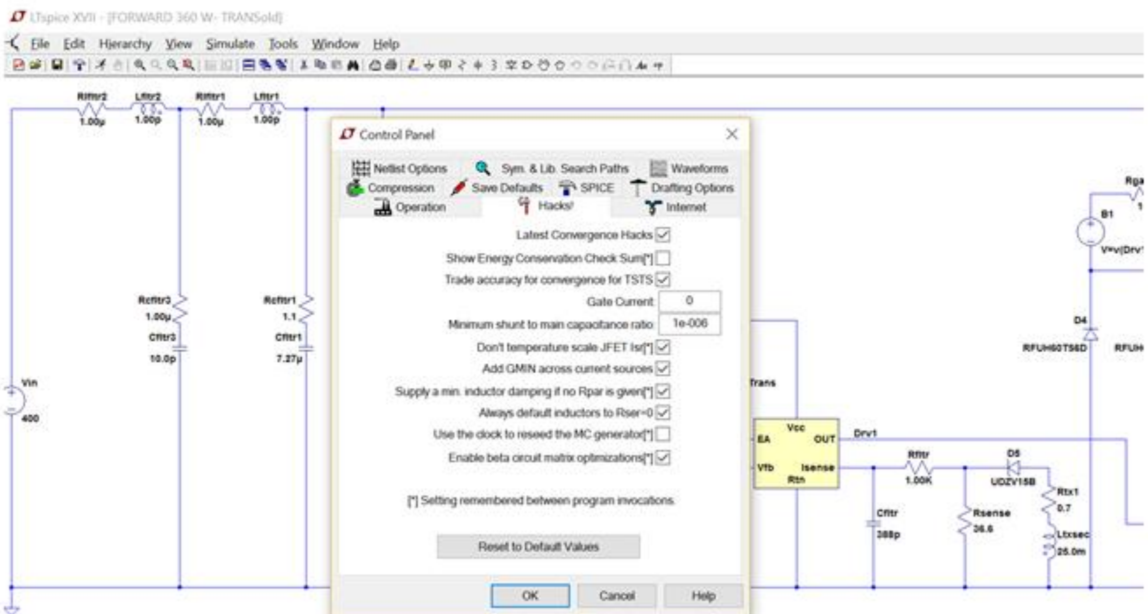
Designers often spend a lot of their development time in creating LTspice® models for their converters. This is a process that requires experience with magnetics models and power controllers. A powerful feature of RidleyWorks® is the ability to automatically generate LTspice® schematic files with just the click of a button.

When you click on the SPICELaunch® icon, you will have the option to create LTspice® schematics, or to adjust loop sweep settings. When using this feature for the first time, click on the button to create the files, and you will see the message that three LTspice® files have been created. These can be found in the folder RidleyWorks LTspice that can be found in the C:\Ridleyworks directory.

5.2 Simulation Settings in LTspice®

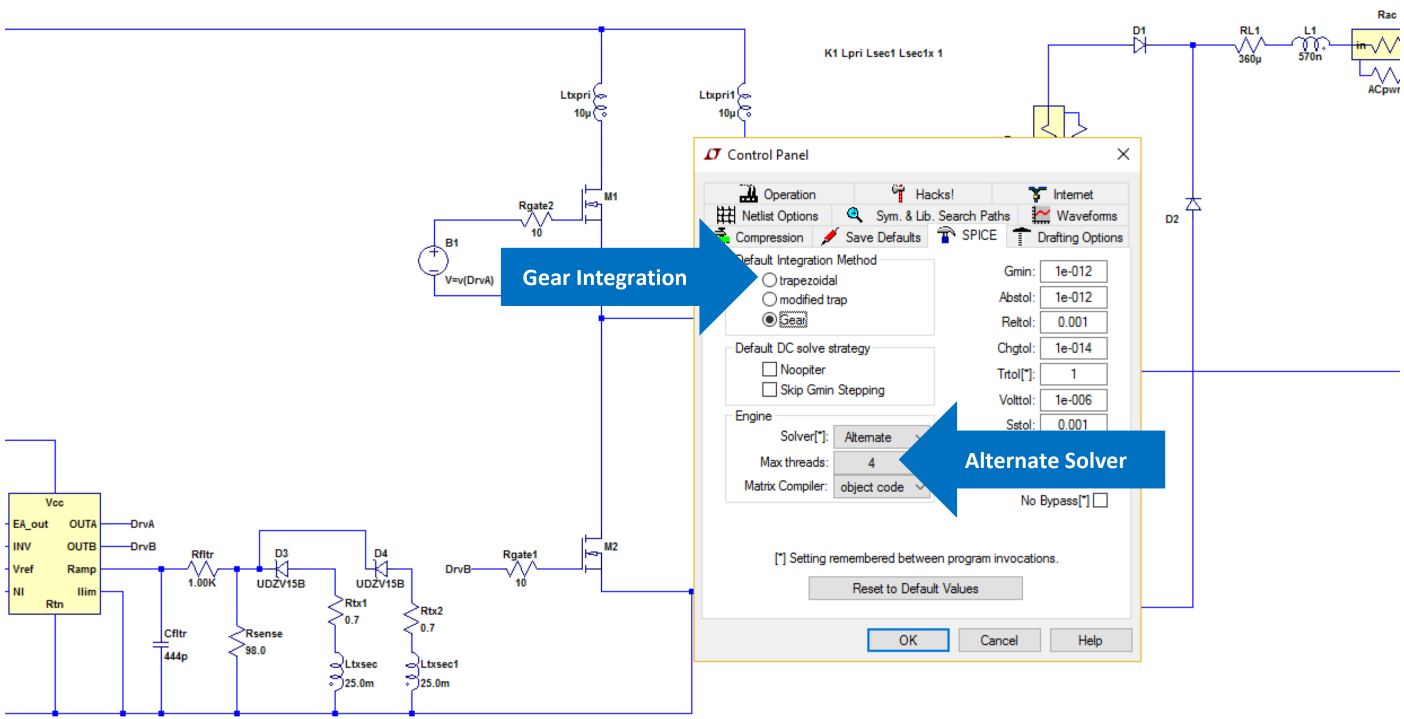
If you double click on the LTspice® TRAN file that is in the folder on your desktop, it will automatically open LTspice® .

Before running a simulation, it is recommended that you set some parameters of LTspice® for better simulation accuracy. Select Tools Control Panel Hacks! and set the check boxes as shown below.



LTspice®, like all versions of spice, may sometimes struggle to converge during a simulation, and you may need to adjust other simulation parameters to help it along. This is one of the advantages of the RidleyWorks® simulation – it never has convergence issues and simulations are almost instantaneous.

Fig. 5.2: Setting Up LTspice® Hacks!



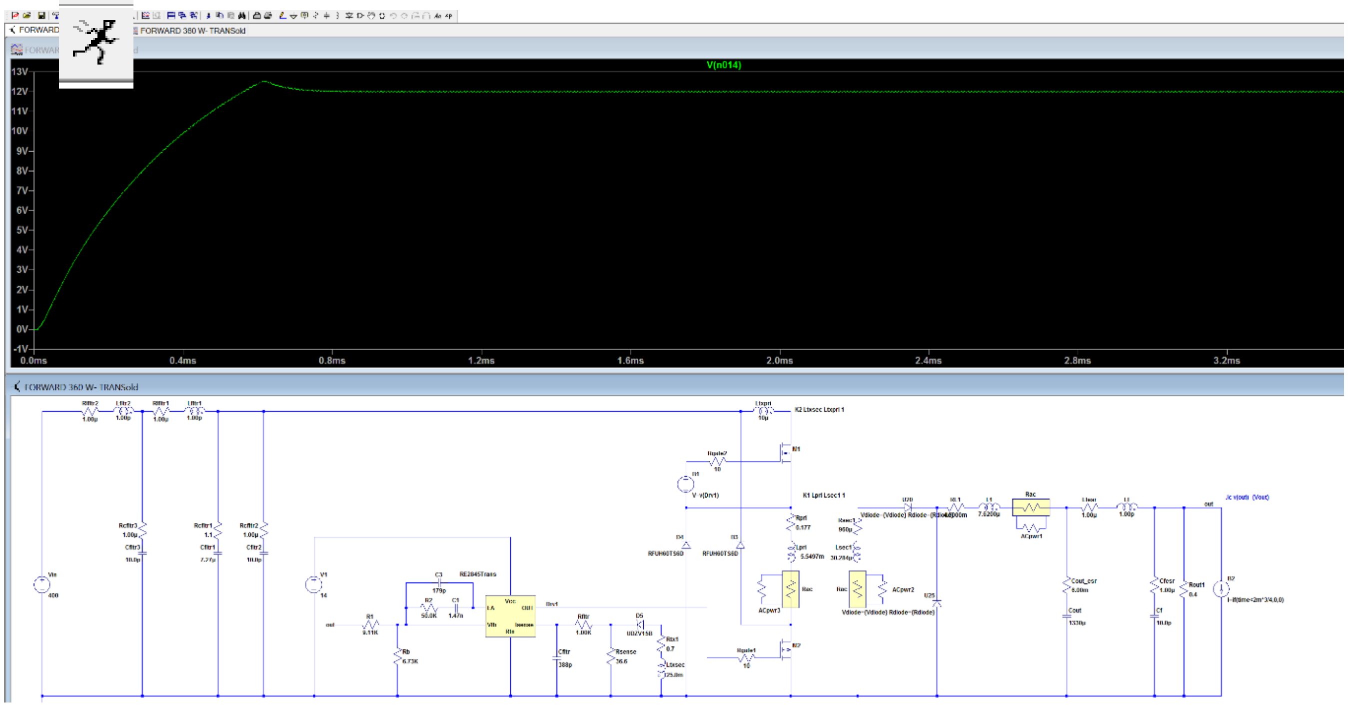
It is also recommended that your Solver Engine should be selected as Alternate. This will give greater simulation accuracy. (The Normal setting will simulate faster which may be useful when generating the Bode plots inside LTspice®.) In some cases, selecting Normal may aid convergence.

We also find that the Gear integration routine works better for transformer isolated converters, but you might want to sometimes try the default settings if you run into difficulty. The default setting is Modified Trap, and that may work better for some circuits.

The integration method settings are not retained in LTspice® each time you exit the program, so you will need to set them each time you come in.

Fig. 5.3: Setting Up LTspice® simulation parameters

5.3 Simulating Transients in LTspice®



When you click on the running man icon, the spice simulation will start. You can then probe any waveform on the circuit. The simulation above is the output voltage during start-up of a circuit. Although the spice simulation may be slow, you have already done all of the design of the control circuit and power parts in RidleyWorks®, and you can now use LTspice® for the things it is good at. You can add any arbitrary circuits, any devices, and move your design much closer to the production schematic.

You will find that the combination of RidleyWorks® and the rapid generation of the schematics for LTspice® changes the way that you design. It should greatly speed up your process.

Fig. 5.3: Running LTspice® Transient Simulations

5.4 Generating Swept Bode Plots in LTspice®

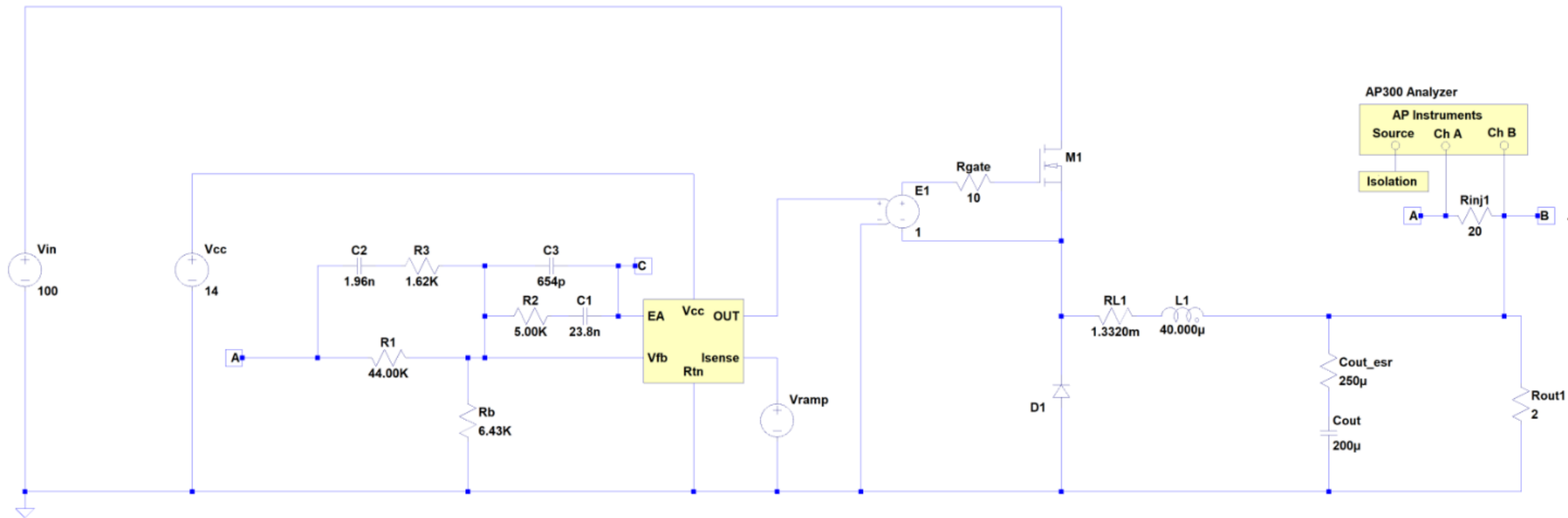


Fig. 5.4: LTspice® Sweep Schematic for Bode Plots

A powerful new feature of RidleyWorks® is the generation of schematics which can produce Bode plots *directly* from the time-domain schematic. There is *no need to simulate small-signal equivalent circuits* which are, by definition, approximations of the real circuit. For the first time, computers (in 2020) are fast enough that the intensive simulation needed for Bode plot generation can be done in a reasonable amount of time. This kind of work was only possible up until now with specialized high-speed simulation programs.

How fast can this be done? At the time of writing this manual, our record for simulation is just 24 seconds for a buck converter. This kind of speed is produced with the following features in place:

1. Automatic generation of a loop compensation that is ruggedly stable

2. Emulation of the features of the AP300 analyzer with frequency-dependent source, automatically configured to give good results.

3. Proper setup of simulation times for steady-state solution.

For the example buck schematic above, the bode generation is initiated by clicking on the running man icon. If you then click on **Point B** of the schematic, you can watch the waveforms being generated ready for measurement. This is done in a very similar way to the powerful AP300 analyzer.

Generation of the sweep schematic is a single button click in RidleyWorks®, and there is no need to change any settings from the default file that is generated. Figure 5.5 shows the repeated sweep waveforms generated in LTspice® . It is very instructional to watch this procedure to enhance your understanding of how the AP300 frequency response analyzer works.

These waveforms are generated by clicking on the Run icon, then clicking a probe on the output voltage of the converter.

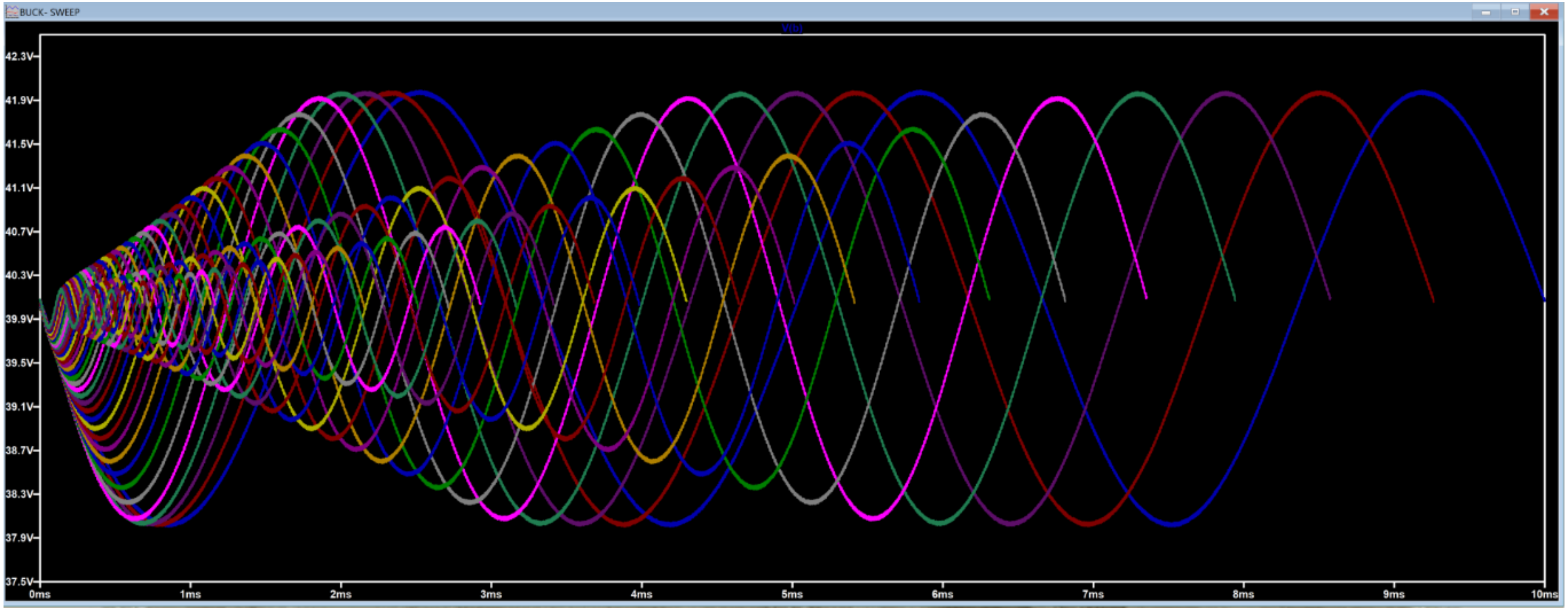


Fig. 5.5: LTspice® Sweep Waveforms at Point B of the Circuit.

Once the sweeps shown above are done, and the data has been processed by LTspice®, click on **View** then **Spice Output Log.** (If the Spice Error Log option is greyed out, LTspice® has not yet finished processing the simulation data.) A file like that shown in Figure 5.6 will then appear:

```
Direct Newton iteration for .op point succeeded.
.step freq=300
.step freq=323.933
.step freq=349.774
.step freq=377.678
.step freq=407.807
.step freq=440.34
.step freq=475.468
.step freq=513.398
.step freq=554.355
.step freq=598.579
.step freq=646.33
.step freq=697.892
.step freq=753.566
.step freq=813.682
.step freq=878.593
.step freq=948.683
.step freq=1024.36
.step freq=1106.08
.step freq=1194.32
.step freq=1289.6
.step freq=1392.48
.step freq=1503.56
.step freq=1623.51
.step freq=1753.02
.step freq=1892.87
.step freq=2043.88
```

Fig. 5.6: LTspice® Error Log

Right click in the space to the right of the numbers on the screen and select the option **Plot .Step'd meas data**. Answer **Yes** to the question that pops up about complex data, and you will see Bode plot Axes appear. If you right click in the plot area, you can then add traces. Specify b/a for Loop Gain, b/c for Power Stage Gain, and c/b for Compensation Gain curves.

Figure 5.6 shows the sweep for a buck power stage gain. Notice that approaching half the switching frequency, noise is apparent on the sweep. This noise characteristic will be very familiar to anyone who has used the AP300 or the RidleyBox on a switching power supply.

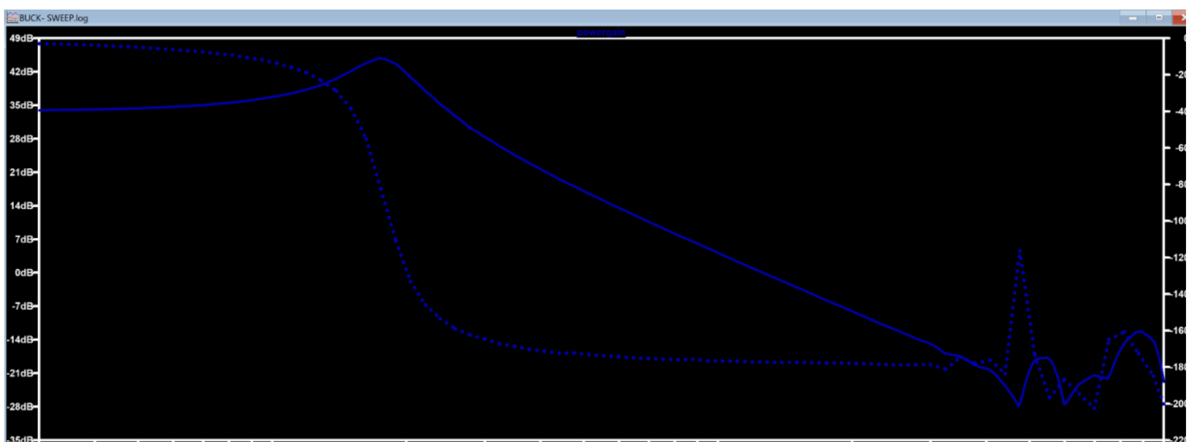


Figure 5.7 shows the sweep for a buck power stage.

Fig. 5.7: LTspice® Bode Plot of a Buck Power Stage Gain

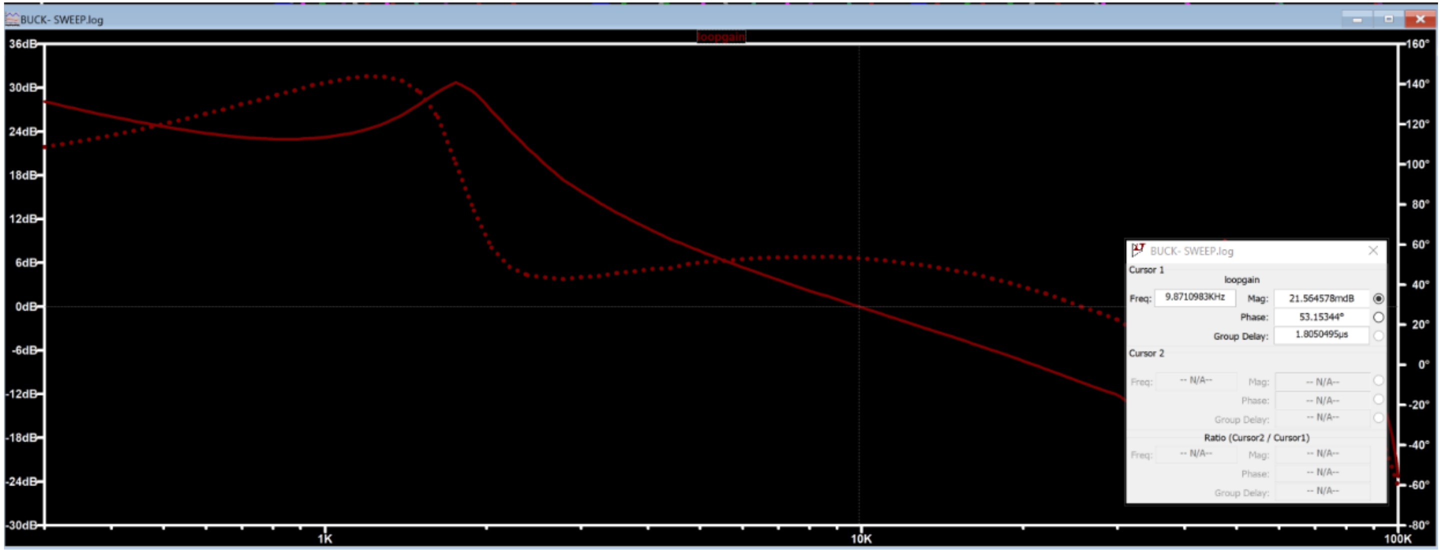


Figure 5.8 shows the sweep for a buck power loop gain. Notice that the phase of this plot is reading phase *margin*, not phase. The cursors on the loop gain show the crossover frequency and the phase margin at this crossover.

Fig. 5.8: LTspice® Bode Plot of a Buck Loop Gain

The ability to sweep the transfer functions on the large-signal schematic inside LTspice® is a powerful tool for finding the bode plots of circuits that don't have small-signal models available for them and we encourage you to explore this feature of the software.

However, there are times when technique this cannot be used, such as:

- The circuit is not stable
- The desired sweep frequency is too low (10 Hz is very time consuming in LTspice®)
- Characteristics of the ac sweep are below the noise-floor resolution of the technique.

The next section of this manual describes another method of obtaining loop gains and other transfer functions via the small-signal models.

5.5 Generating Small-Signal Bode Plots in LTspice®

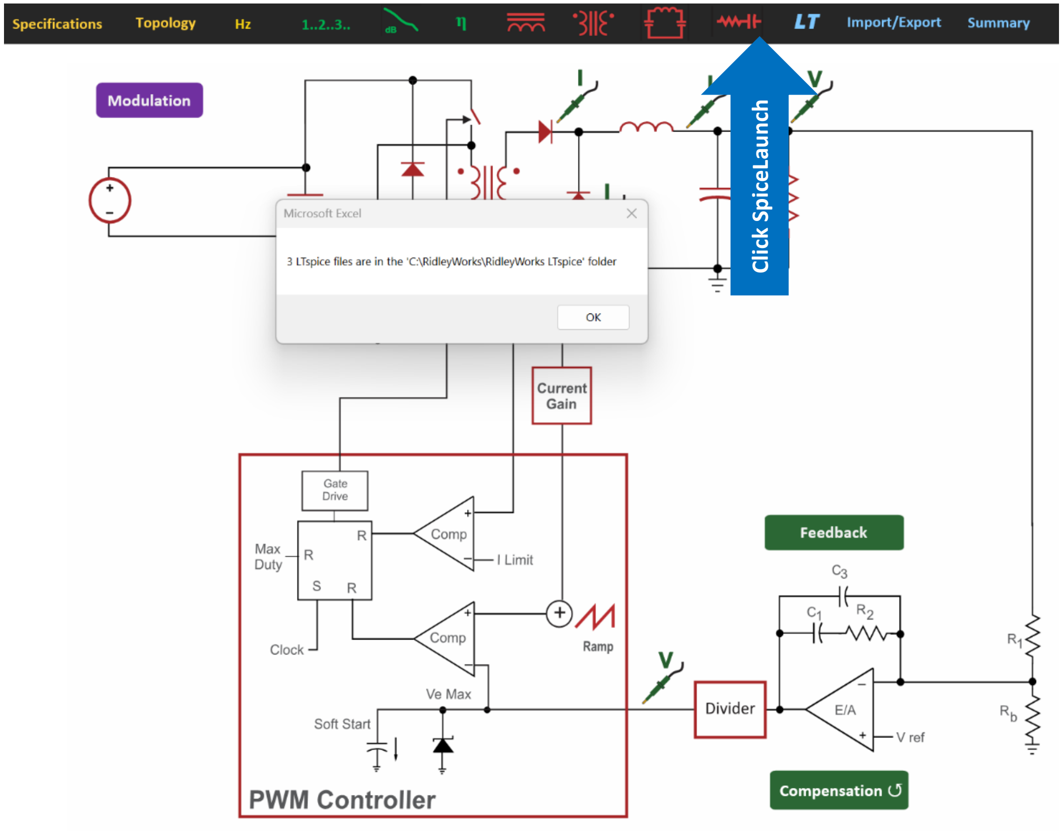


Fig. 5.9: Exporting an LTspice® Small-Signal File. Three files are automatically exported: a transient simulation, a swept-sine simulation, and a small-signal model.

The major modern work on small-signal models for power converters was initiated by Dr. Vatché Vorpérian with the publication of the PWM Switch Model in 1988. Models were completed by Dr. Ray Ridley with the publication of his Current-Mode Model in 1990. If you are new to power supply design, these models can take some time to become familiar with and implement in your circuit simulator. We have fully automated this process for the first time, putting small-signal analysis at every engineer's fingertip.

Release 12.35 of RidleyWorks® exports a small-signal model for use with LTspice®. This provides a rapid high-speed design and simulation environment for the feedback networks. All characteristics of input and output filters will be seen in the loop characteristics, providing more detailed sweeps than those available in RidleyWorks®.

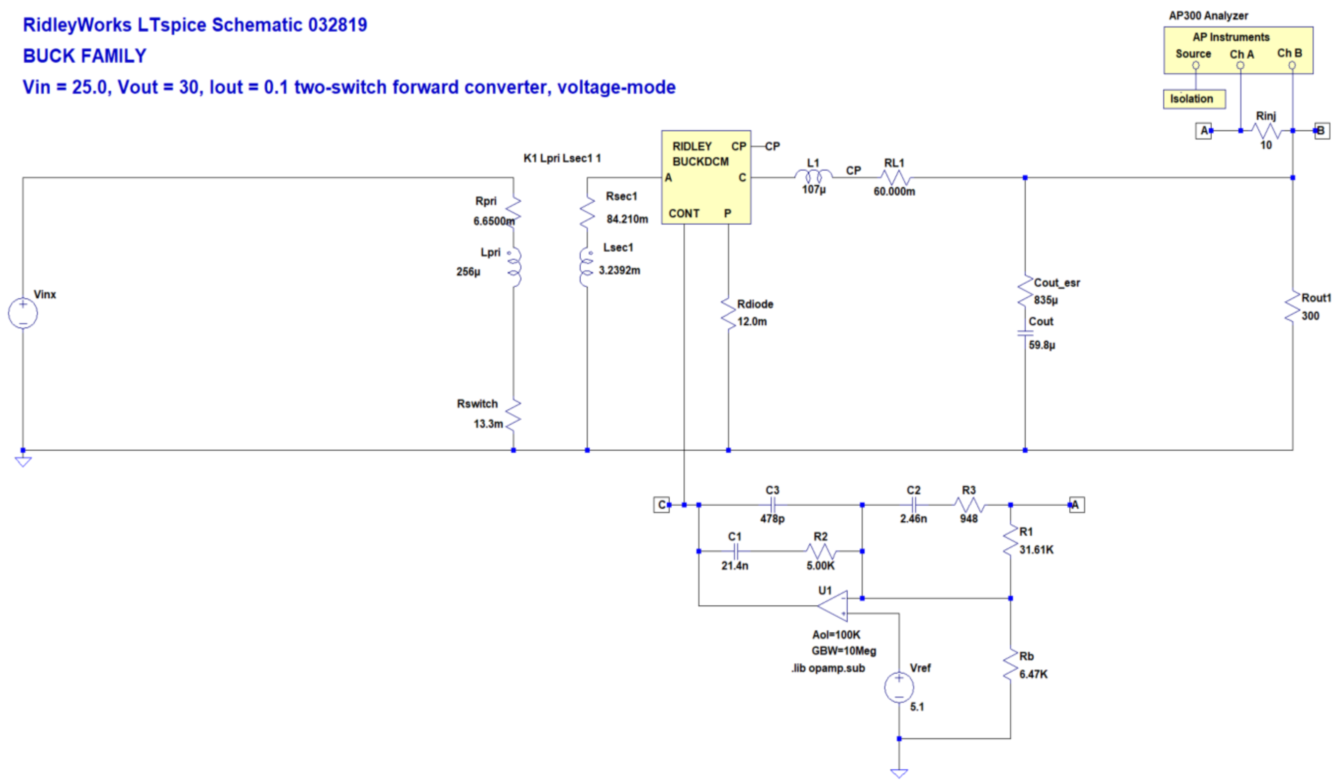


Fig. 5.9: Buck Family Small-Signal Model Exported to LTspice®

Figure 5.5 shows the buck family configuration of the small-signal model in LTspice®. There are three fundamental families that cover all of the topologies in RidleyWorks® - the buck, boost, and buck-boost. Each of these circuits contains a single small-signal element, in this case the Ridley BuckDCM. When the file is exported to LTspice®, RidleyWorks® determines whether the circuit is going to be analyzed in CCM operation or DCM operation.

Notice that the model is invariant for both voltage-mode and current-mode control. This is very important – our small-signal model is the only one that works in both regions. As the compensating ramp for current mode is increased, it is possible to get a system that operates in between voltage-mode and current-mode, but the model gives the right result regardless.

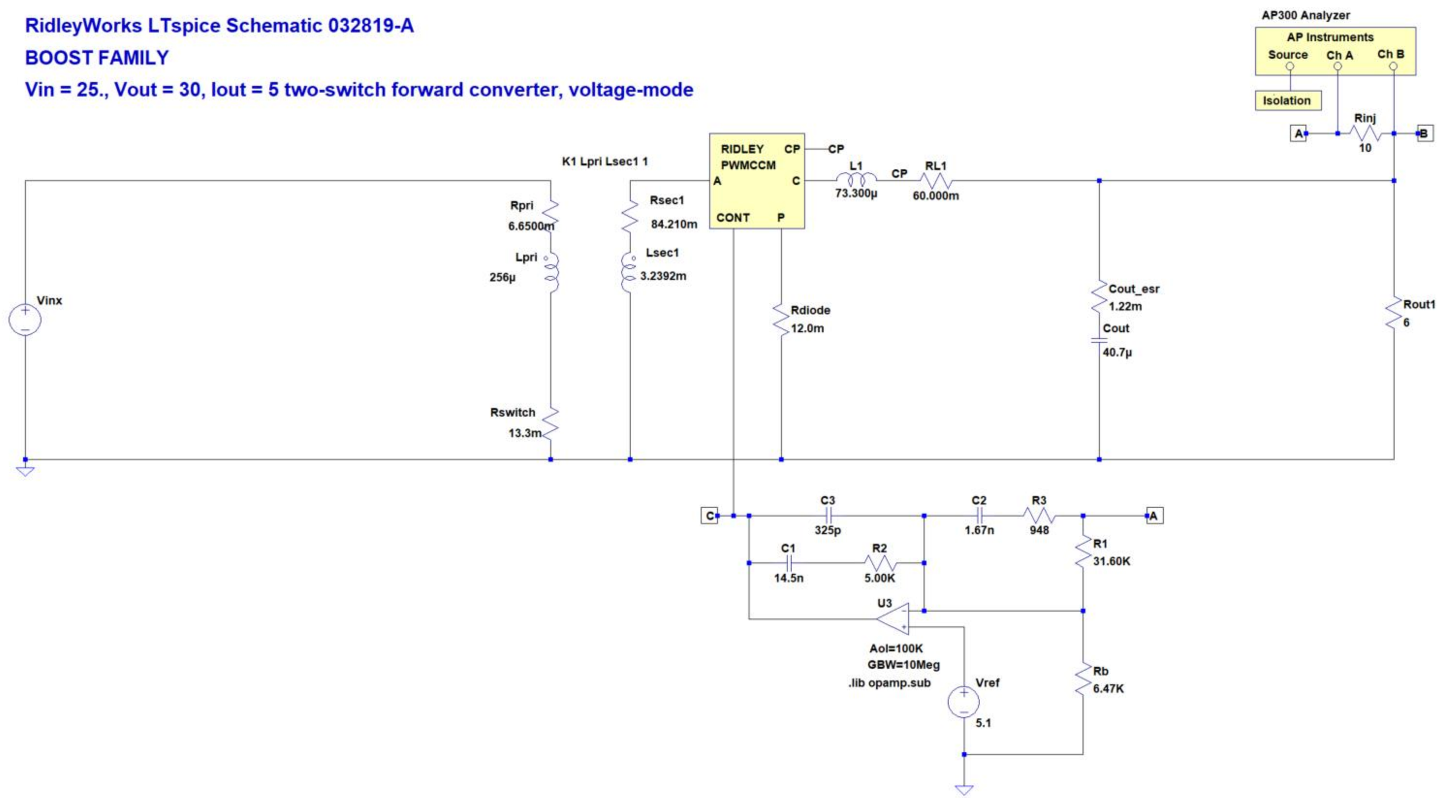


Fig. 5.10: Boost Family Small-Signal Model Exported to LTspice®. Notice the PWM block is the same subcircuit as for the buck converter family.

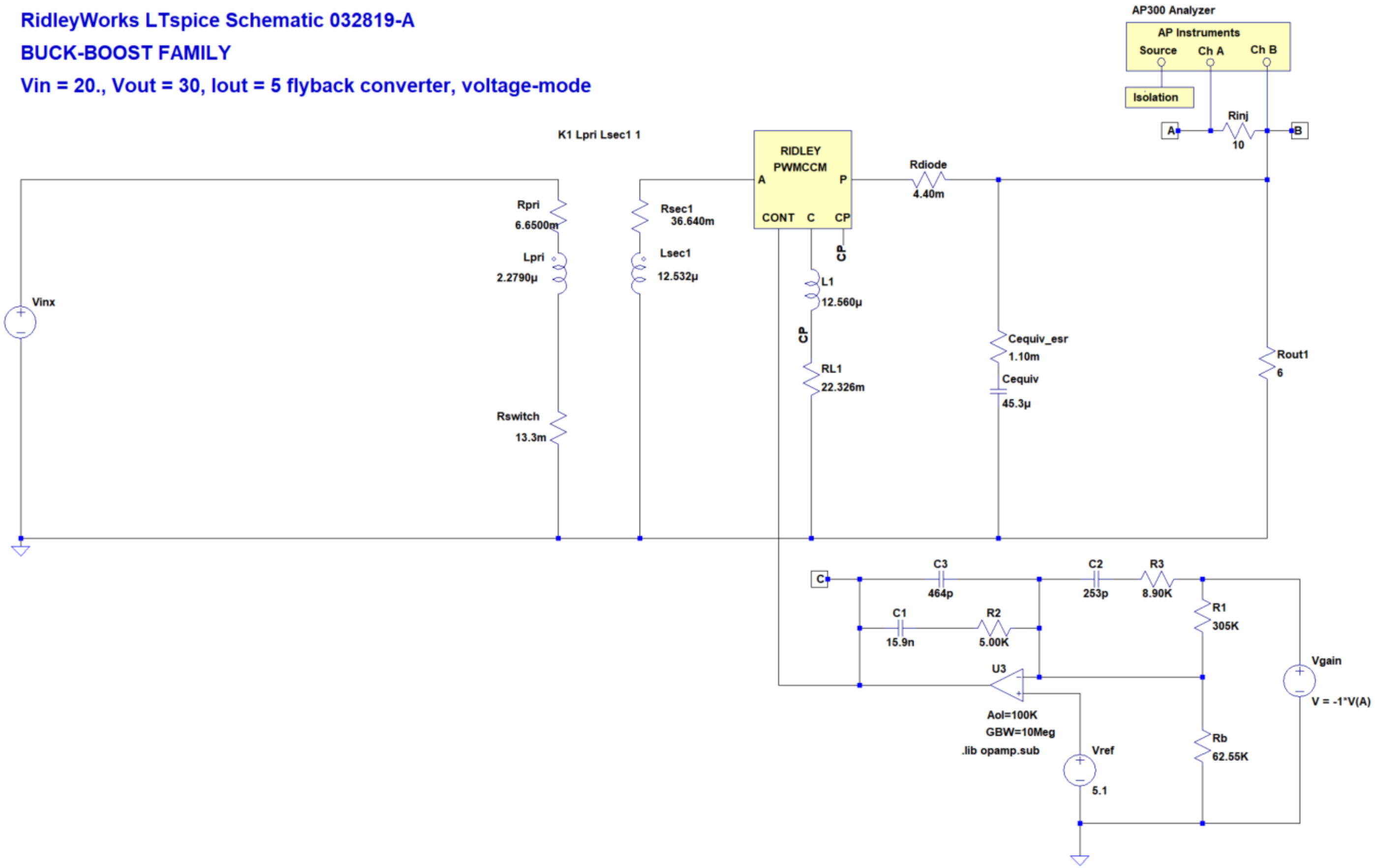


Fig. 5.11: Buck-Boost Family Small-Signal AC Model Exported to LTspice®. Notice the inversion of the output voltage signal in the feedback path.

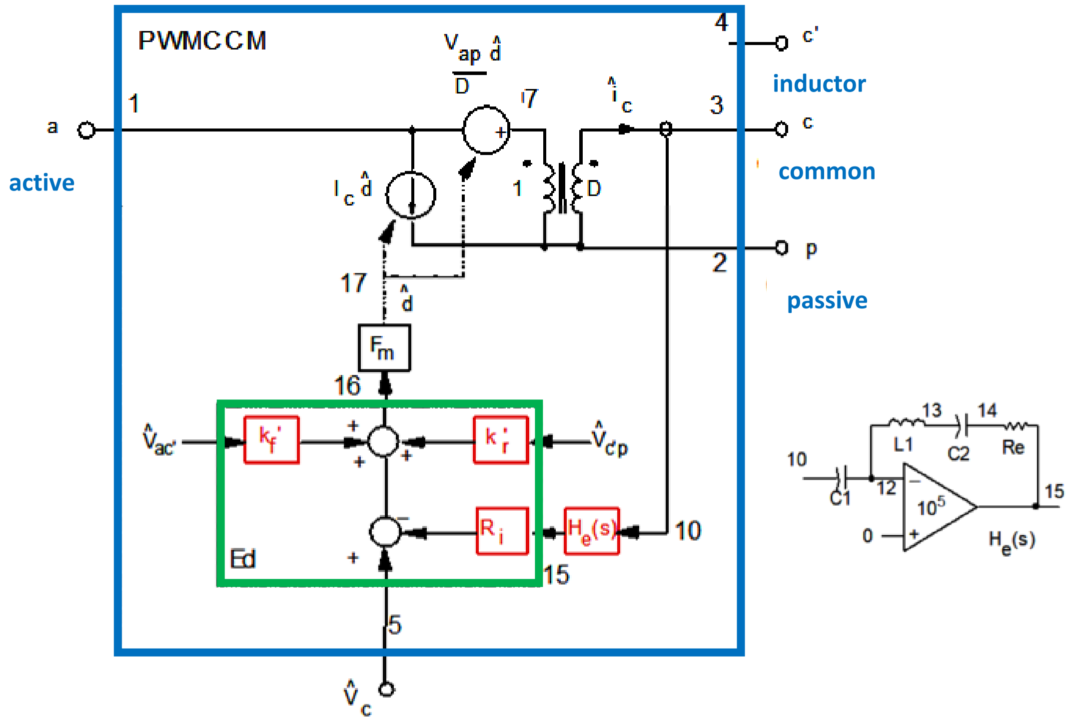


Fig. 5.12: Invariant Small-Signal Subcircuit for Current- or Voltage-Mode control in CCM. For pure voltage-mode control, the gain blocks inside the green rectangle are all zero.

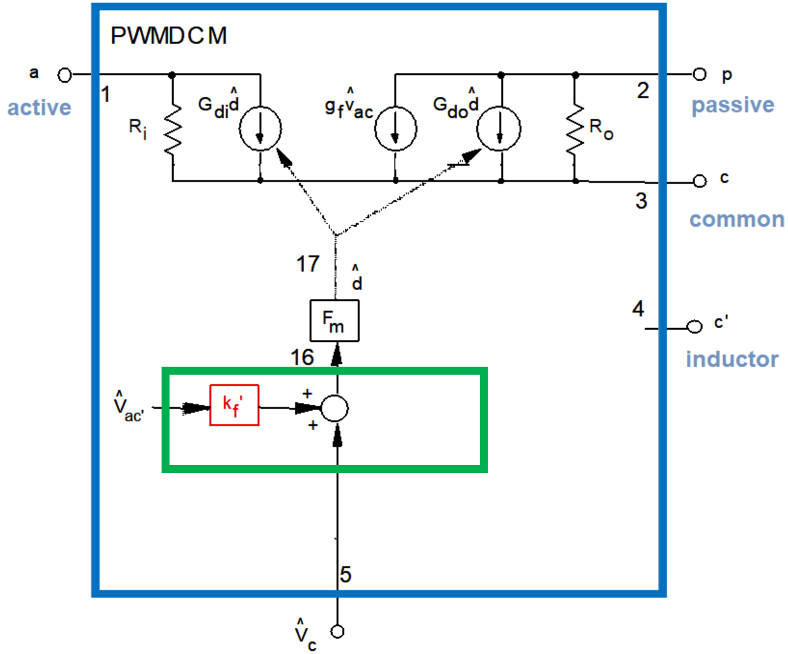
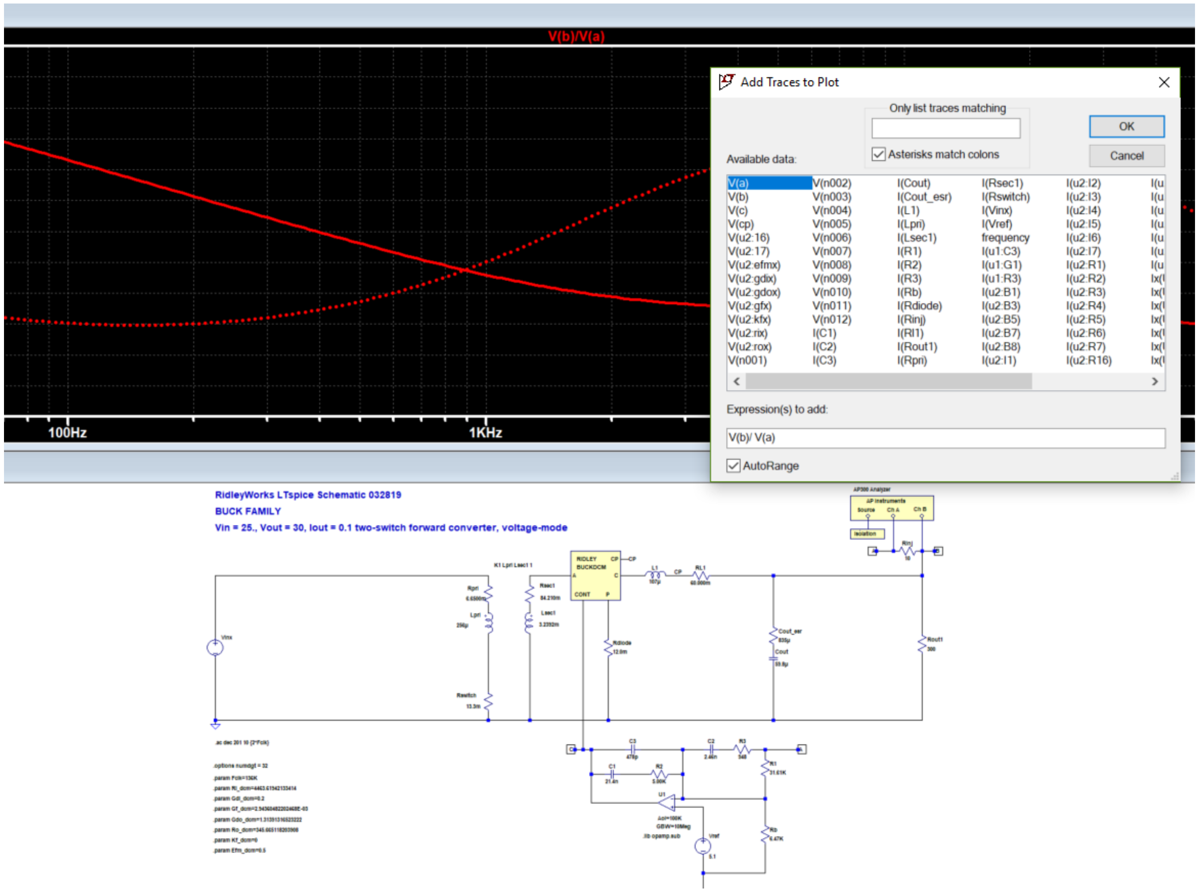


Fig. 5.13: Invariant Small-Signal Subcircuit for Current- or Voltage-Vode Control in DCM. For pure voltage-mode control, the gain block inside the green rectangle is zero.



The figure above shows the loop gain plot for a buck-family converter. Running the sweep only takes a couple of seconds. You can then add any quantities to like to see desired transfer functions. The following are the most useful:

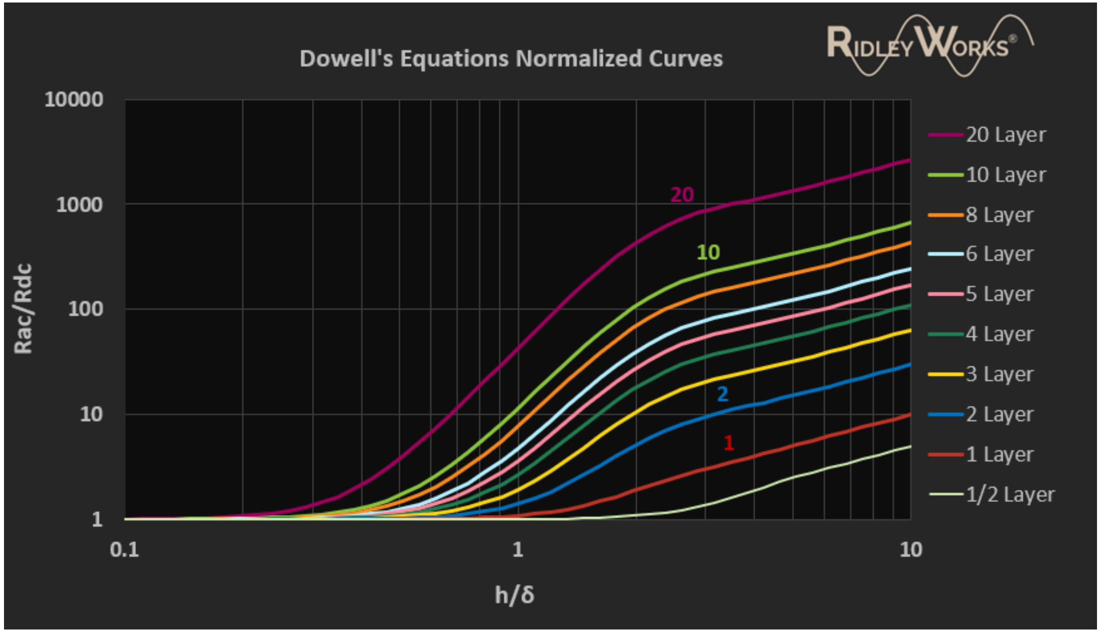
V(b)/V(a) Loop Gain and Phase Margin

V(b)/V(c) Power Stage Gain (specify **-V(b)/V(c)** for the buck-boost family

-V(c)/V(a) Compensation Gain

Fig. 5.14: Plotting the loop gain of a converter consists of just clicking on run, then right click on the plot area. The quantity v(b)/v(a) will be the loop gain and phase MARGIN.

6 Advanced Proximity Loss and Core Loss

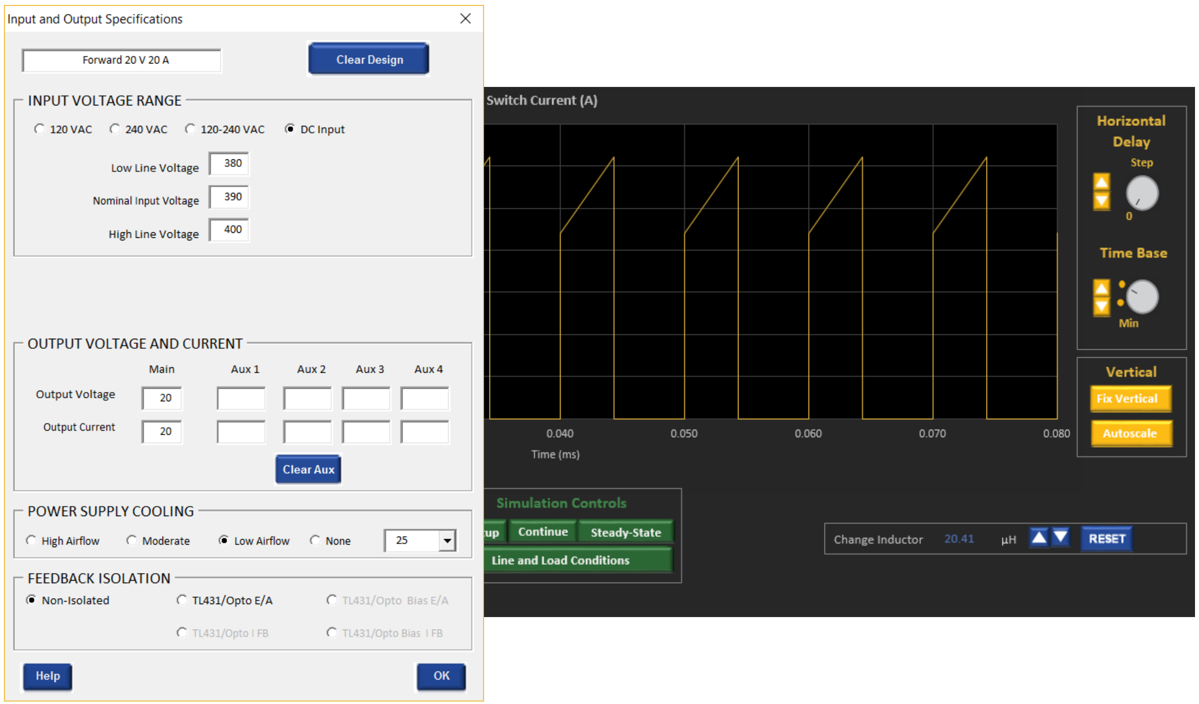


One of the biggest sources of error in designing a power supply is underestimating the losses in inductor and transformer windings. To get a reasonable estimate of how much loss there will be, it is necessary to apply Dowell's equations, a set of highly complex formulae that is usually the domain of PhD students in power electronics. It is estimated that less than 1% of practicing engineers ever use these techniques in their design work.

Fortunately, RidleyWorks® will do this for you. It does this in two ways – first by solving the Dowell's equation at the switching frequency to finding the increase in resistance. The winding current is then broken down into two parts – the dc current, and the ac current, which is the sum of all the harmonics. For a simple approximation, RidleyWorks® assumes that all the ac current is at this frequency. This will give a low estimate of the proximity loss, but it does provide good and fast guidance for how to organize the winding layers and winding size.

Secondly, to get more accurate loss in the windings due to *all* the current harmonics, RidleyWorks® first does a sweep of the Dowell's equation solutions from 1 kHz to 10 MHz. It then generates a circuit network that produces the same impedance versus frequency as Dowell's equations. This network is exported to an LTspice® file where the proximity losses can be directly simulated in the time domain. This process is described in this chapter.

6.1 Estimating Proximity Losses in RidleyWorks®



The specifications for a 100 kHz two-switch forward converter are shown in the Specifications entry form for RidleyWorks®. This is for a 20 V, 20 A isolated converter running from a high-voltage input.

Figure 7.1 shows the simulated primary current for the converter. The rms primary winding loss, using the dc value of the resistance of the winding is calculated to be 0.491 W. This is the value that most designers would use for estimating temperature rise, but we will soon see that it is much too low.

Fig. 6.1: Steady-State primary current from RidleyWorks® Simulation

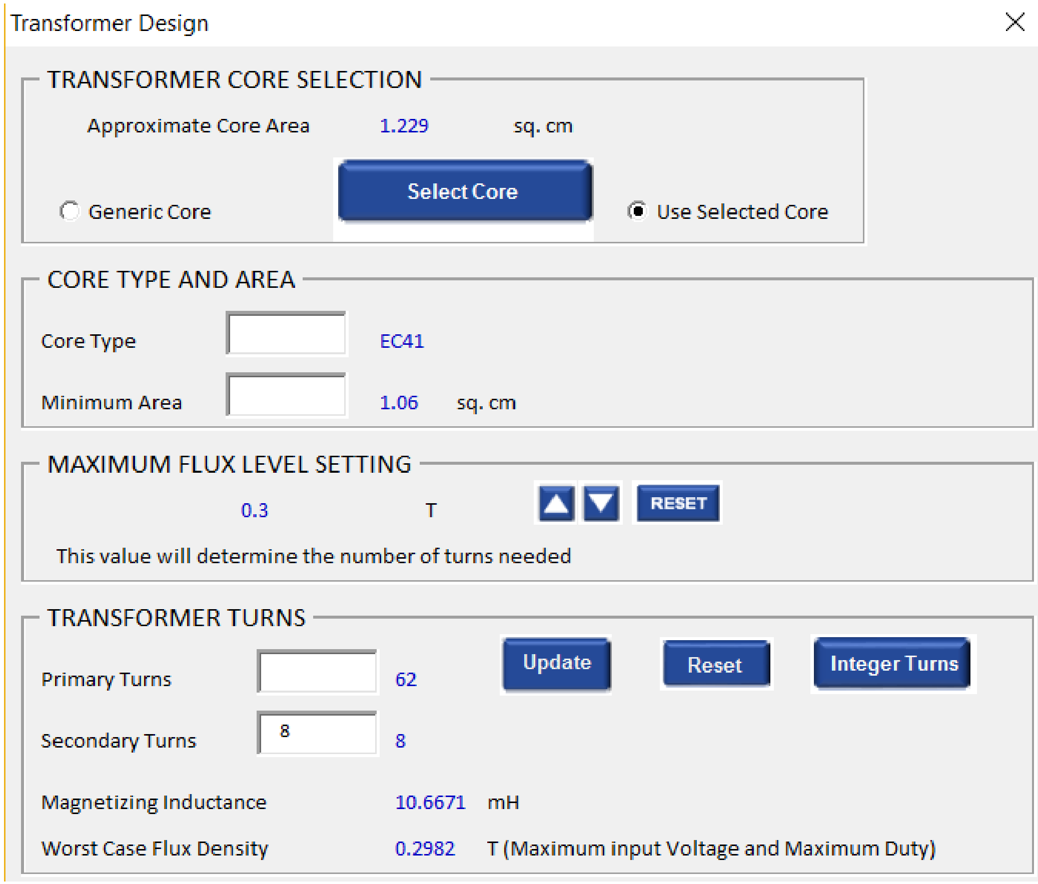


Figure 6.2 shows the important transformer parameters using and EC41 core. There are 62 primary turns, and 8 secondary turns.

Fig. 6.2: Transformer parameters from RidleyWorks®.

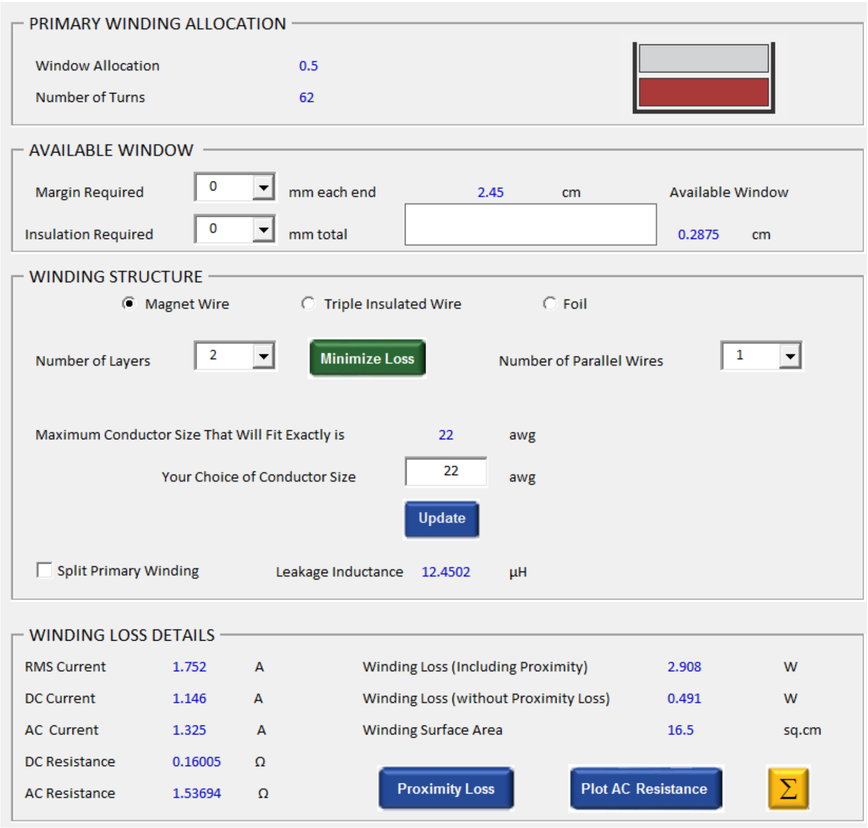


Fig. 6.3: Primary winding structure and analysis from RidleyWorks®

Figure 6.3 shows the structure and analysis of the primary winding of the transformer. Two layers of 22 awg wire are used to give a dc resistance of 160 mOhm. When this is multiplied by the square of the rms current, we see 0.491 W of loss.

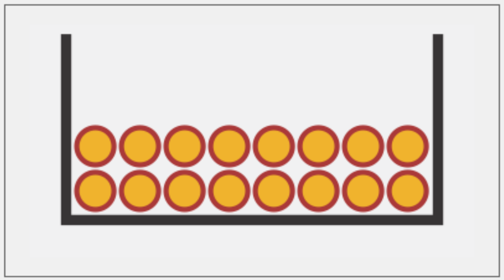
However, the ac resistance of the winding, calculated at 100 kHz, is much higher at 1.54 ohms, almost 10 times the dc resistance. RidleyWorks® takes the rms of the ac current squared and multiplies by this number, adds to the dc current squared times the dc resistance, to get the total losses. You can see that they are much higher at 2.91 W. This is an increase of four times.

This more detailed and insightful information allows you to experiment with different numbers of layers, wire sizes, and parallel strands to see what gives the lowest loss combination. The **Minimize Loss** button automatically tries different layer counts to find the best solution for you.

Winding Layer Height 7.28534879 cm

Skin Depth at 100000 Hz 0.024 cm

Each Layer is 3.01908156 Skin Depths



PROXIMITY RESISTANCE MULTIPLIER BY LAYER

1	2
3.0295	16.176

Overall AC/DC Resistance Multiplier 9.602914

DC Winding Resistance 0.16005 Ω

AC Winding Resistance 1.53694 Ω

Total power dissipation in a winding is given by :

$$P_d = b_w \sum_{i=1}^{n} l_i \frac{1}{h_i \eta_i \sigma} H_i^2 \left[\left(1+\alpha_i^2\right) G_{1_i} - 4\alpha_i G_{2_i} \right]$$

The H field is calculated from

$$H_i = \frac{N_i I_i}{b_w} \qquad b_w = winding\ width$$

Complex functions are needed to calculate the losses :

$$G_{1_i} = \Delta_i \frac{\sinh 2\Delta_i + \sin 2\Delta_i}{\cosh 2\Delta_i - \cos 2\Delta_i}$$

$$G_{2_i} = \Delta_i \frac{\sinh \Delta_i \cos \Delta_i + \cosh \Delta_i \sin \Delta_i}{\cosh 2\Delta_i - \cos 2\Delta_i}$$

The ratio of the winding layer height to skin depth is

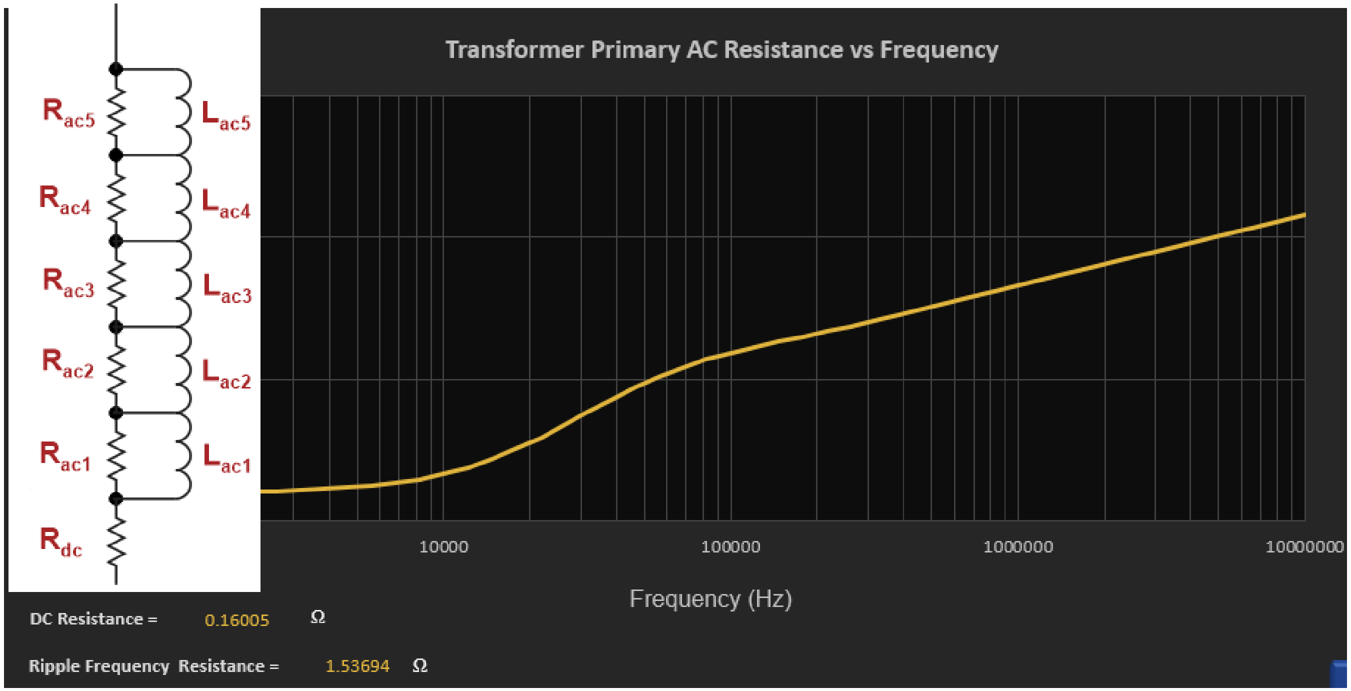
$$\Delta_i = \frac{h_{cu_i}}{\delta} \qquad skin\ depth\ \delta = \sqrt{\frac{2}{\omega \mu_o \sigma \eta}}$$

$$\sigma = conductivity \qquad \mu_o = 4\pi\ x10^{-7} \qquad \eta = porosity$$

If you click on the blue **Proximity Loss** button, you can see a detailed breakdown of the resistance of the wire layer-by layer. The complexity of Dowell's equations shows why few engineers ever attempt this analysis. Fortunately, RidleyWorks® automates the whole procedure.

Fig. 6.4: Primary loss analysis by layer together with Dowell's equations published in 1966.

6.2 Advanced Proximity Losses with LTspice® Circuit Modeling



Once you have defined the winding structure and are happy with the total dissipation, you can begin the process of LTspice® model generation and simulation. When you click on this button

RidleyWorks® will proceed to sweep the solutions to Dowell's equations from 1 kHz to 10 MHz and generate an equivalent circuit model to match the complex impedance.

This process must be followed for each of the magnetics windings to generate the equivalent circuit models to be used in LTspice® . You can then click on the button below to export the complete circuit model.

Fig. 6.5: Sweep of ac resistance and equivalent circuit model

6.3 Running LTspice® for Proximity Loss

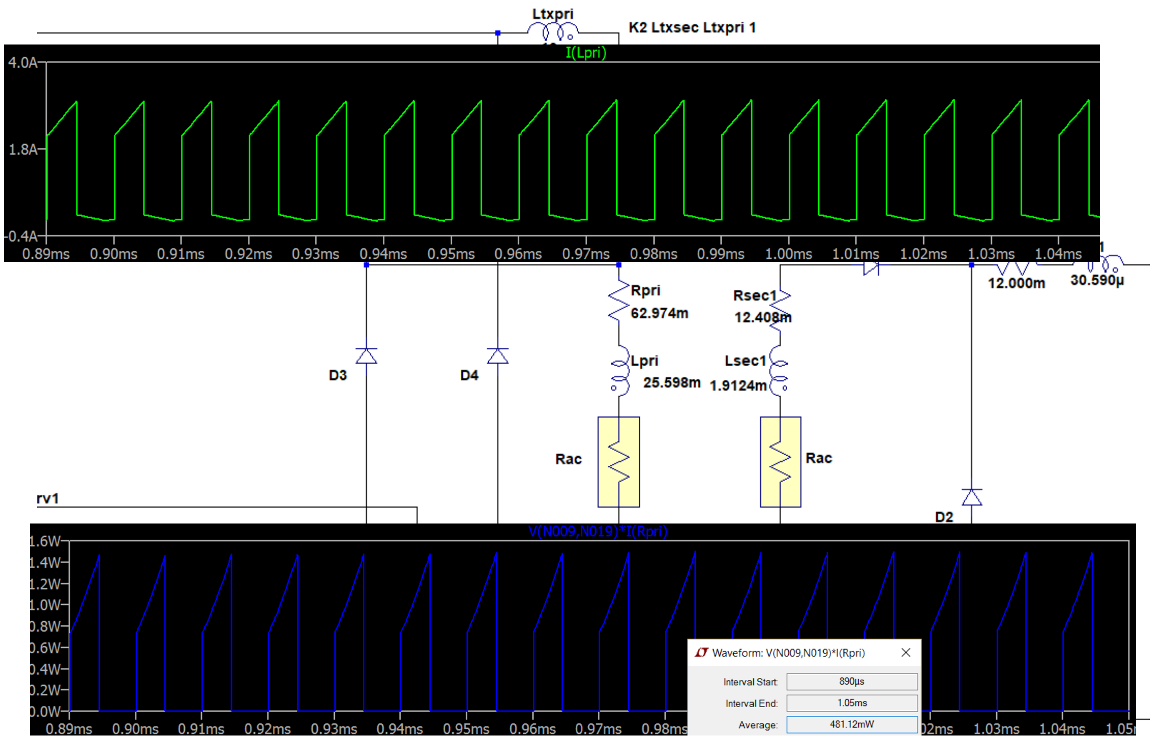


Fig. 6.6: Conventional dissipation in transformer primary (no proximity loss).

Figure 6.6 shows the two-switch forward schematic in LTspice® . The yellow resistive elements are the ac proximity models for each of the windings.

Step 1 is to simulate the converter to steady-state and plot the primary switch current **I(Lpri)** as shown.

You can plot the conventional dissipation in the primary (without proximity) by holding down the **ALT** key and clicking on **Rpri**. This results in the waveform shown in Figure 6.8. If you click on the name of the waveform in blue, while holding down the **Ctrl** key, it will show the average dissipation. You can see that this number agrees with the dissipation in RidleyWorks® before proximity is added.

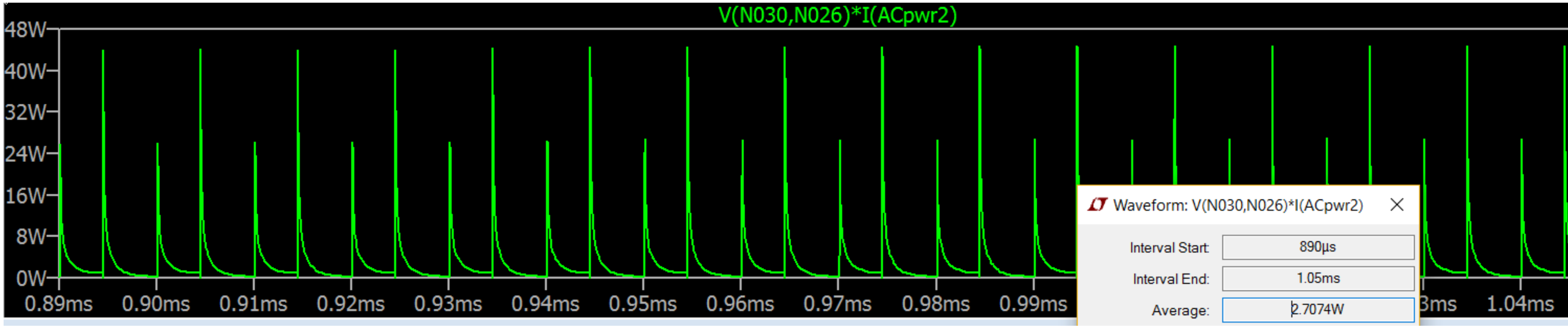


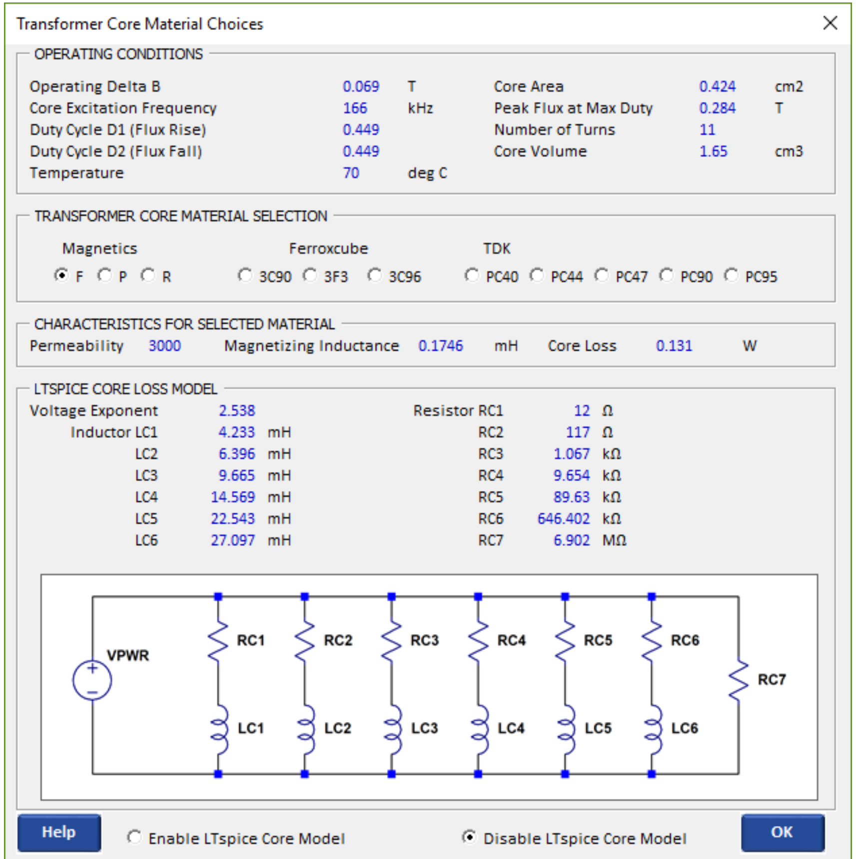
Fig. 6.7: Additional proximity loss dues to ac resistance model.

You can now plot the additional proximity loss dissipation in the primary by holding down the **ALT** key and clicking on the circuit element **Rac** in the primary. This results in the waveform shown in Figure 6.9. If you click on the name of the waveform in green, while holding down the **Ctrl** key, it will show the average dissipation. The sum of the conventional dissipation and the **Rac** dissipation is the total winding loss in the primary.

You will probably see the total dissipation predicted by LTspice® is higher than that predicted by RidleyWorks®. The LTspice® number will be more accurate since it includes the dissipation in the resistors at higher harmonic frequencies. Depending on the topology and operating point, the difference may be considerable. In the example above, the conventional losses are 0.481 W. The additional proximity losses produce an extra 2.7 W of dissipation!

Once you have mastered this technique, you will be doing proximity loss calculations that are more advanced than just about anyone in the industry. Attempting to do this work without the aid of RidleyWorks® is extremely time-consuming and difficult – you now have a powerful tool that gives you a huge design advantage.

6.4 Advanced Core Losses with LTspice® Circuit Modeling

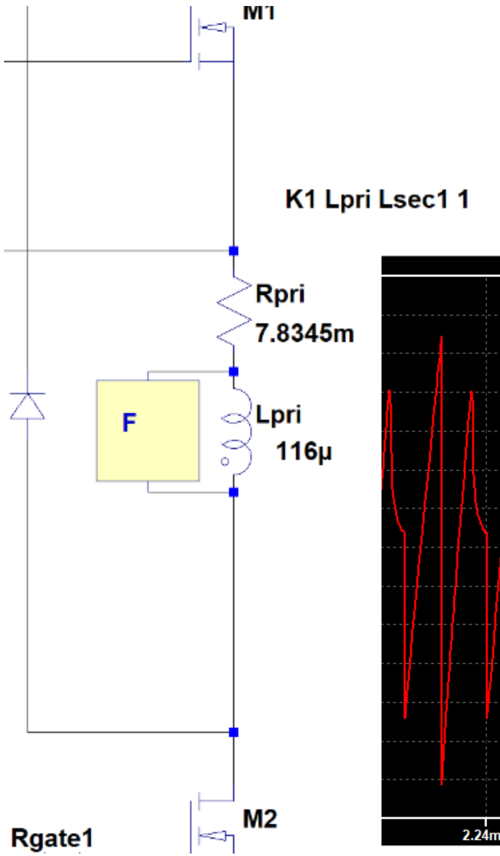


With the selected core shape and material, RidleyWorks® will automatically calculate parameters for core loss simulation in LTspice®.

The component elements of the core loss model define how the core will behave with different drive frequencies and amplitudes. This is a unique feature of RidleyWorks®.

Fig. 6.8: Selecting a core material will produce a matching LTspice® circuit model

You can plot the instantaneous core losses of the block to the left of the transformer primary by holding down the **Alt** key and clicking on the block. The instantaneous core loss can be averaged by holding down the **Ctrl** key and clicking on the name of the waveform shown in red below. You will observe that for the same flux swing in a transformer, the losses increase with a shorter duty cycle, as is observed empirically. You will also see that the losses of a transformer decrease with switching frequency if the turns are left unchanged on the core.



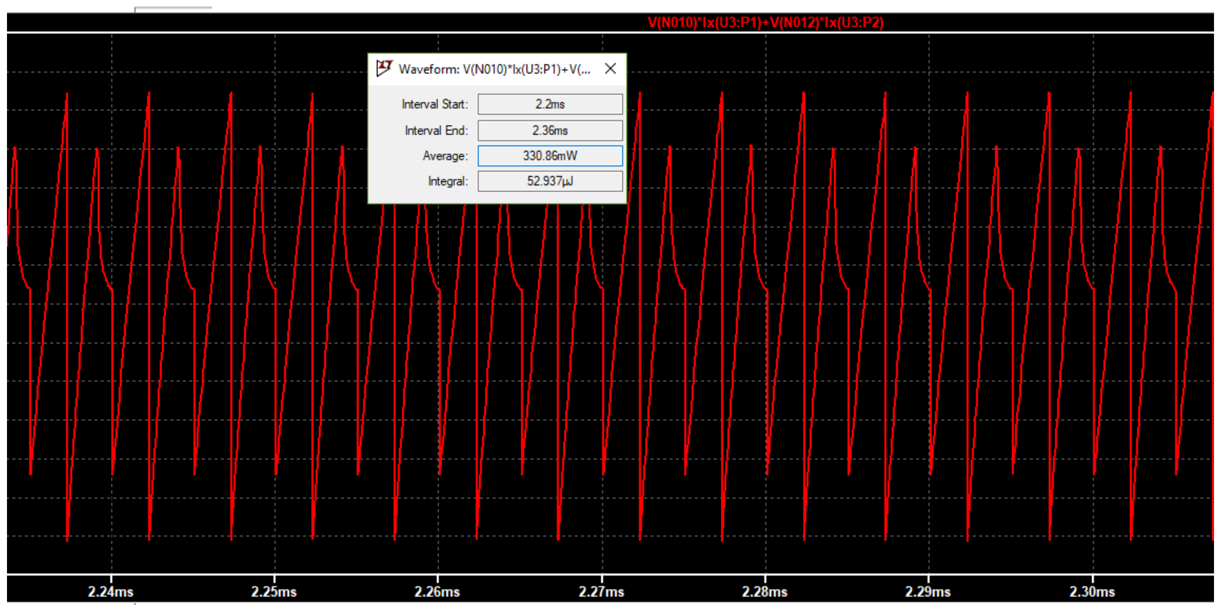


Fig. 6.9: Core loss model parameters are automatically loaded into your LTspice® schematic. Instantaneous core losses are shown, together with the average value.

7 Using RidleyWorks® to the AP310 Analyzer

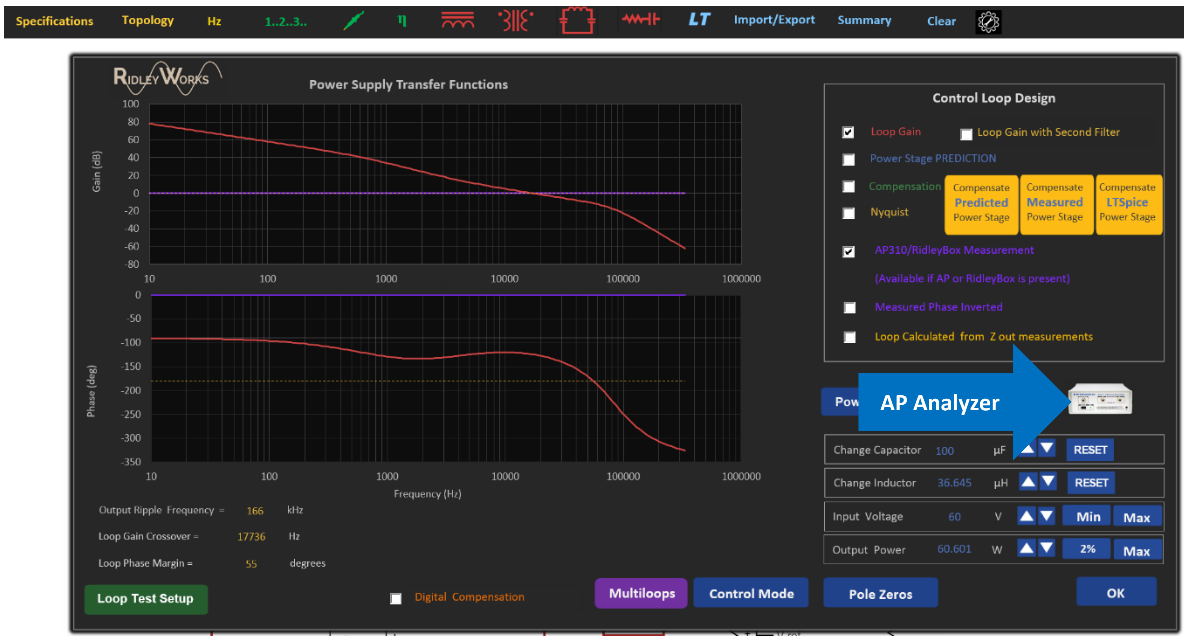


Fig. 7.1: RidleyWorks Opening page with AP310 and RidleyBox® Icons

7.1 Connecting RidleyWorks® to the AP

RidleyWorks® has a powerful and user-friendly interface to the AP310 frequency response analyzer. This enables you to control complex analyzer functions and settings with a single button click inside RidleyWorks®.

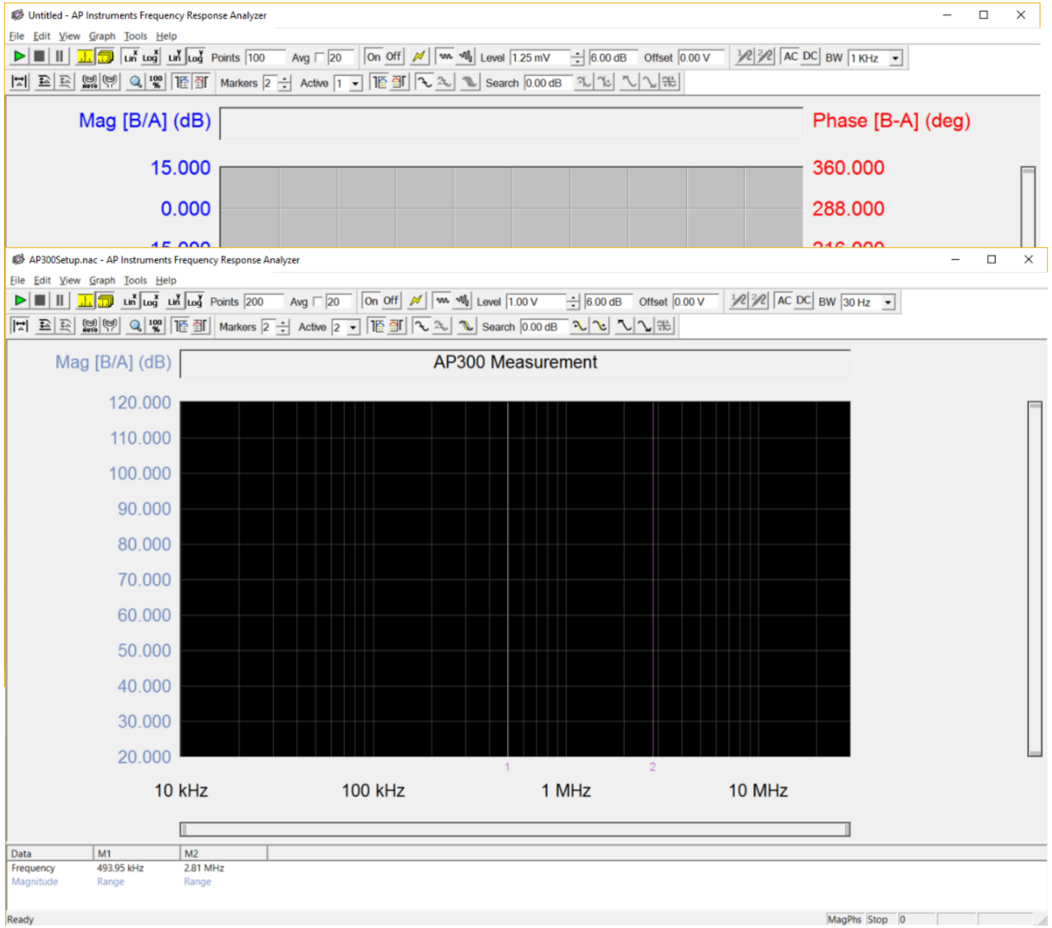
When you click on the loop option of the schematic, you will see the opening screen shown here. Click on the AP310 analyzer picture to start controlling your instrument. (You can also click on the AP analyzer on any page that you see within RidleyWorks®.

RidleyWorks® will connect to the AP310 Measurements page shown below.

Fig. 7.2: AP Instruments Analyzer Interface Page

This page provides a powerful interface for driving the AP300 and collecting multiple data sweeps. There are 8 preset settings on this page for magnetics measurements, control measurements, impedance measurements, and PSRR measurements.

First, turn on the power switch of the AP analyzer that you have, and click on the Connect button to automatically launch the APfra software.



Once you see the screen to the left, Click on File>Open Setup and select AP300Setup.nac to load new default settings. Notice that the sweep is now logarithmic, and the stop frequency is 30 MHz with a drive signal of 1 V.

Fig. 7.3: APfra default opening screen and screen after loading AP300Setup file.

7.2 Measuring Transfer Functions

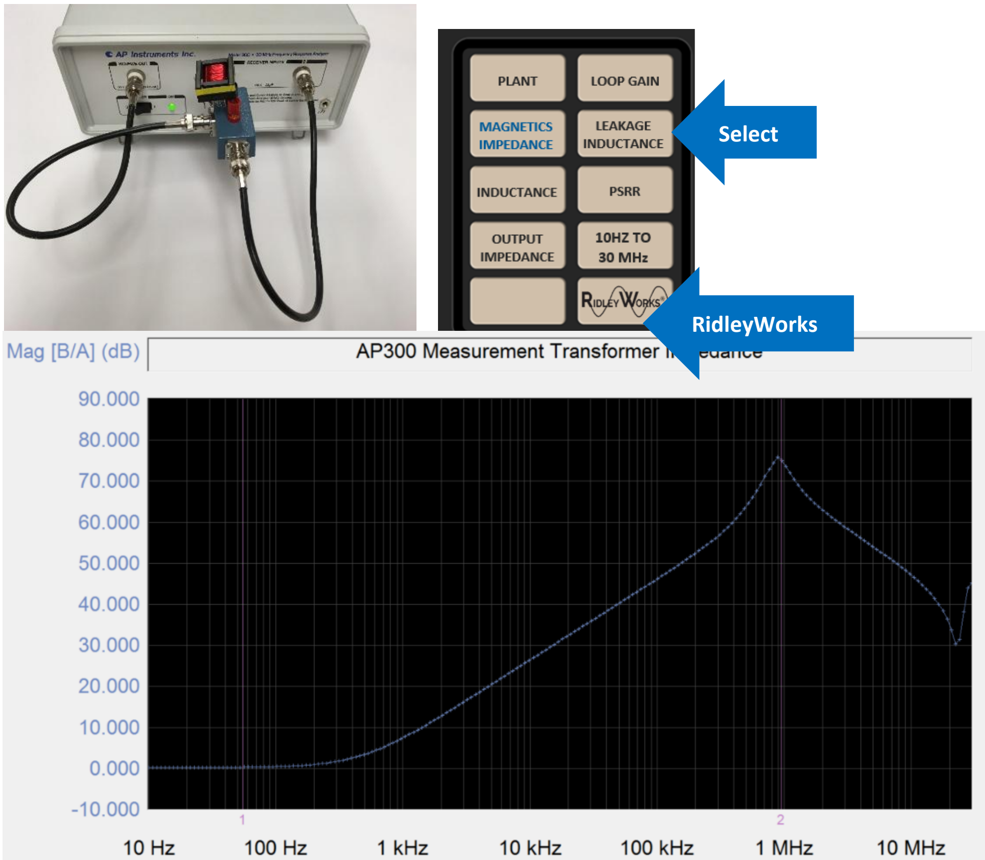


Fig. 7.4: Magnetics impedance measurement setup and test results

There are multiple setups for automating the settings on the AP310. This includes measuring the power stage plant, loop gains, magnetic impedances, PSRR and output impedance.

Each of these setups will adjust the source, bandwidth, and frequency range for the AP analyzer. You can still adjust these quantities individually within the APfra software as needed.

Full details of these measurement setups are provided in the AP310 analyzer user manual.

8 Using RidleyWorks® with the RidleyBox®

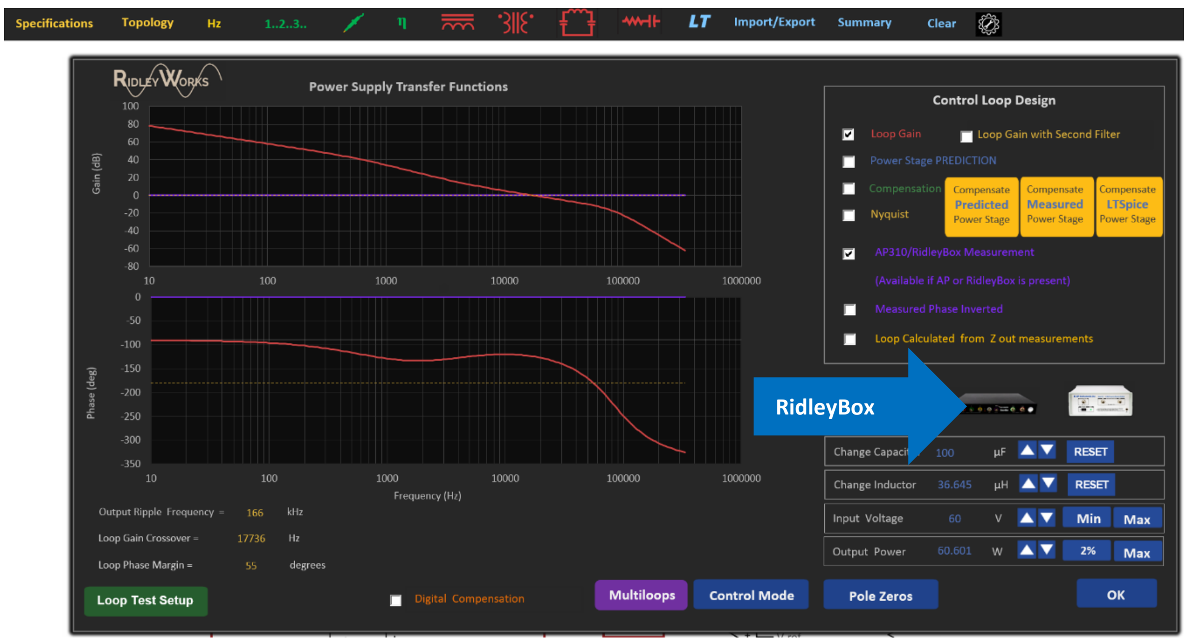


Fig. 8.1: RidleyWorks® Opening page with AP310 and RidleyBox® Icons

8.1 Connecting RidleyWorks® to the RidleyBox®

RidleyWorks® has a powerful and user-friendly interface to the RidleyBox®. This enables you to control complex analyzer functions and settings with a single button click inside RidleyWorks®.

When you click on the loop option of the schematic, you will see the opening screen shown here. Click on the RidleyBox® picture to start controlling your instrument. (You can also click on the RidleyBox® icon on any page within RidleyWorks® where it appears.)

RidleyWorks® will connect to the RidleyBox® measurements page shown below.

You can also connect to the RidleyBox with the COOL-I-O icon in the Toolbar. COOL-I-O

Fig. 8.2: RidleyBox® Interface Page

8.2 RidleyBox® Interface

This page provides a powerful interface for driving the RidleyBox® and collecting multiple data sweeps. There are 8 preset settings on this page for magnetics measurements, control measurements, impedance measurements, and PSRR measurements.

The first step is to turn on the RidleyBox® from within the software. No physical power button is needed. When you click on the button, you will soon hear a message that the connection is complete, and the power indicator will turn green.

You can then select the measurement that you want to make. Parameters will be taken from RidleyWorks® to set up each individual measurement properly. Please refer to the RidleyBox® manual for more details.

9 Designing LLC Converters with RidleyWorks®

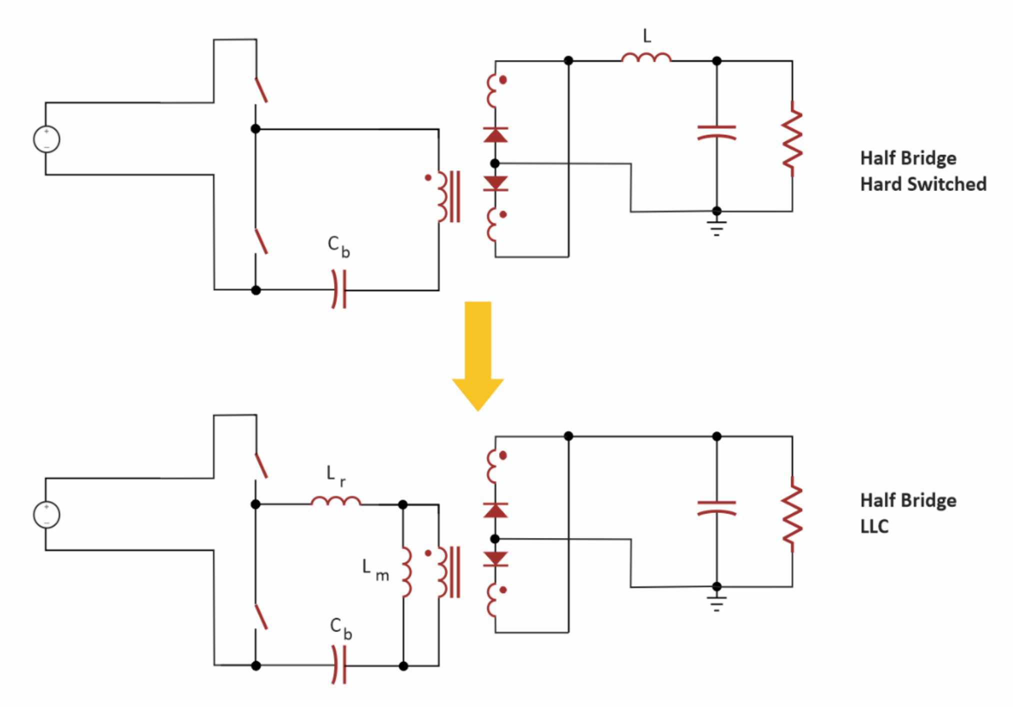


Figure 10.1: Comparison of Half-Bridge and LLC Circuits

9.1 Why LLC?

The half-bridge converter has always held a place in the world of practical power conversion. Now imagine a technology that:

1) Eliminates the output inductor
2) Reduces the AC coupling cap by an order of magnitude
3) Achieves zero-voltage (lossless) switching in the primary
4) Achieves zero-current switching in the secondary

That is what the LLC (Inductor-Inductor-Capacitor) converter achieves. It has been around for decades, but only really became widespread after regulations mandated power factor correction at the input of the power converter.

The LLC is not a panacea for all applications. Drawbacks include:

1) Poorly understood circuit design process (solved by RidleyWorks®)
2) Higher peak and RMS currents
3) Complex and widely-varying control characteristics
4) Light-load regulation challenges
5) Complex magnetics design process
6) Unsuitable for wide input ranges

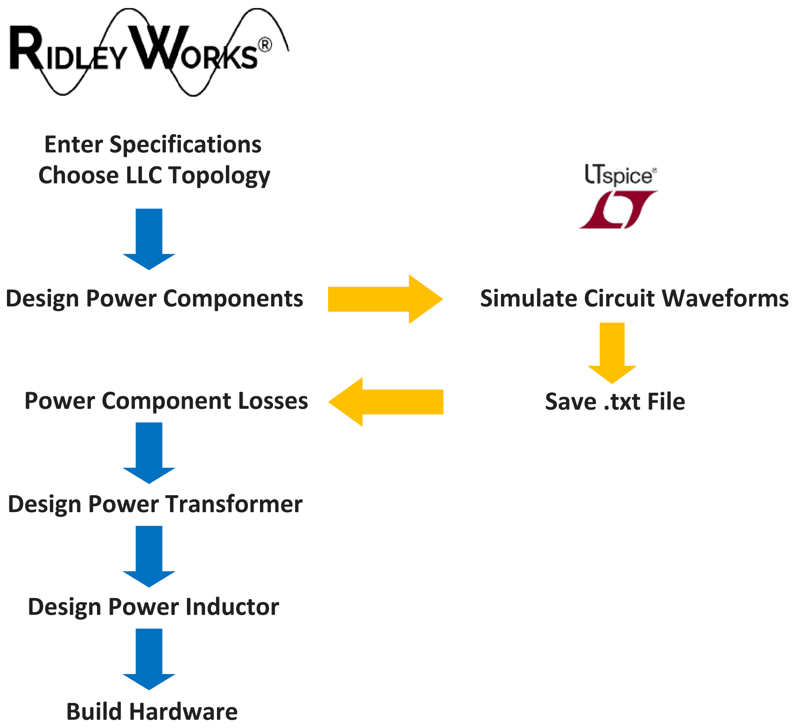


Figure 10.2: RidleyWorks® LLC Design Process Overview

9.2 RidleyWorks® LLC Design Overview

RidleyWorks® has a fully automated power stage design algorithm built into the program. The design process is based upon the superb work of Sanjaya Maniktala, and the follow-on work by Nicola Rosano. Their basic philosophy is that you only need one design optimized for an LLC converter, and every other converter specification will take this optimized design and scale it according to voltage and current specifications. There is no need to reinvent the wheel every time you design an LLC converter.

The overview of the design flow is shown in Figure 9.2. Once you have designed the power components you can then run a circuit simulation in LTspice® to verify the circuit waveforms under different operating conditions. These waveforms can then be brought back into RidleyWorks® for more detailed component design work.

Less experienced designers may want to let their preferred magnetics vendor design the custom parts needed for your converter. You just need to provide the specified component values and simulated waveforms.

Experienced designers usually like to have full control over the magnetics design and parts specification. RidleyWorks® will help you with this process. It can work at a basic level, or deep into issues of magnetics proximity core losses if you prefer.

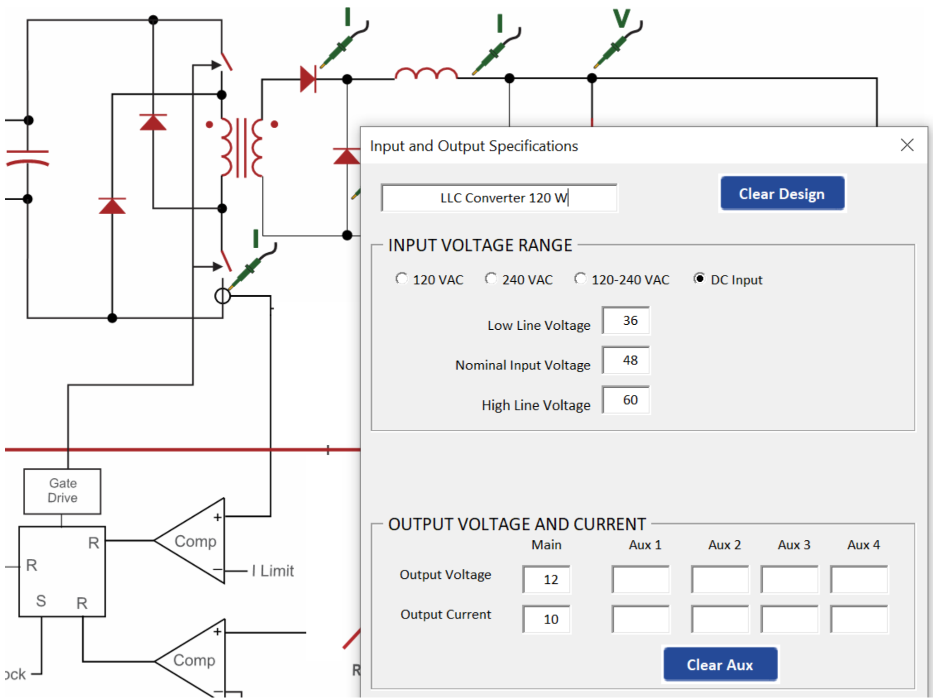


Fig. 9.3: RidleyWorks® Specification Form

9.3 Entering Specifications

The first step of the design is to enter the input line range and the output load specifications. A narrow range of input voltage will lead to a more optimal design for the LLC converter. The best application of LLC is when you have a pre-regulated rail, as would be provided by a power factor correction circuit on the input to the converter.

When you click OK, you will see the form on the next page allowing you to specify the switching frequency for your design.

9.4 LLC Frequency Selection

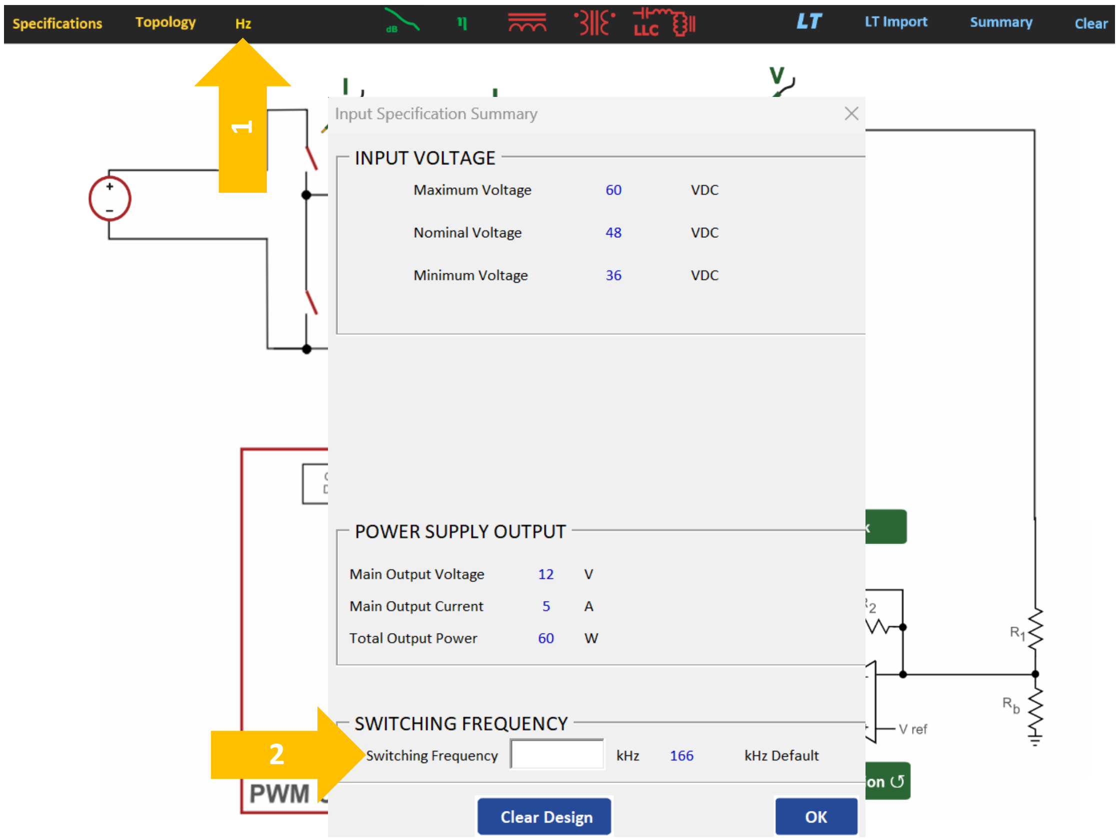


Fig. 9.4: Frequency Selection for the LLC Converter

Click on the Hz icon to open the form to enter the desired frequency of the converter.

For the LLC converter, the switching frequency entered on this form is the target frequency for full-load, low-line operation. At higher line input, or lighter load, the regulated frequency of operation will be higher than this target frequency. This is a natural characteristic of the LLC converter.

9.5 LLC Topology Choices

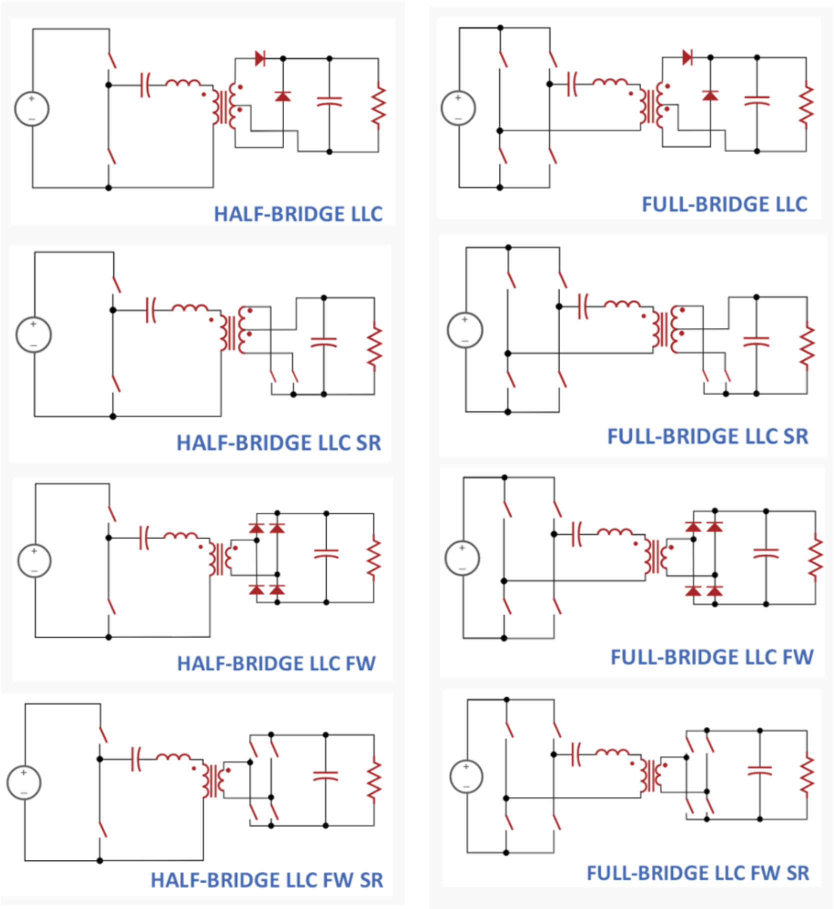


Fig. 9.5: LLC Circuit Variations Available in RidleyWorks®

The first step is to click on the Topology button and select either full-bridge or half-bridge LLC converters. You can then select the variation that you want to use for your design.

Unlike other topologies, the design of the circuit elements is not automatic. The next section explains how to proceed with the design.

9.6 LLC Circuit Optimization

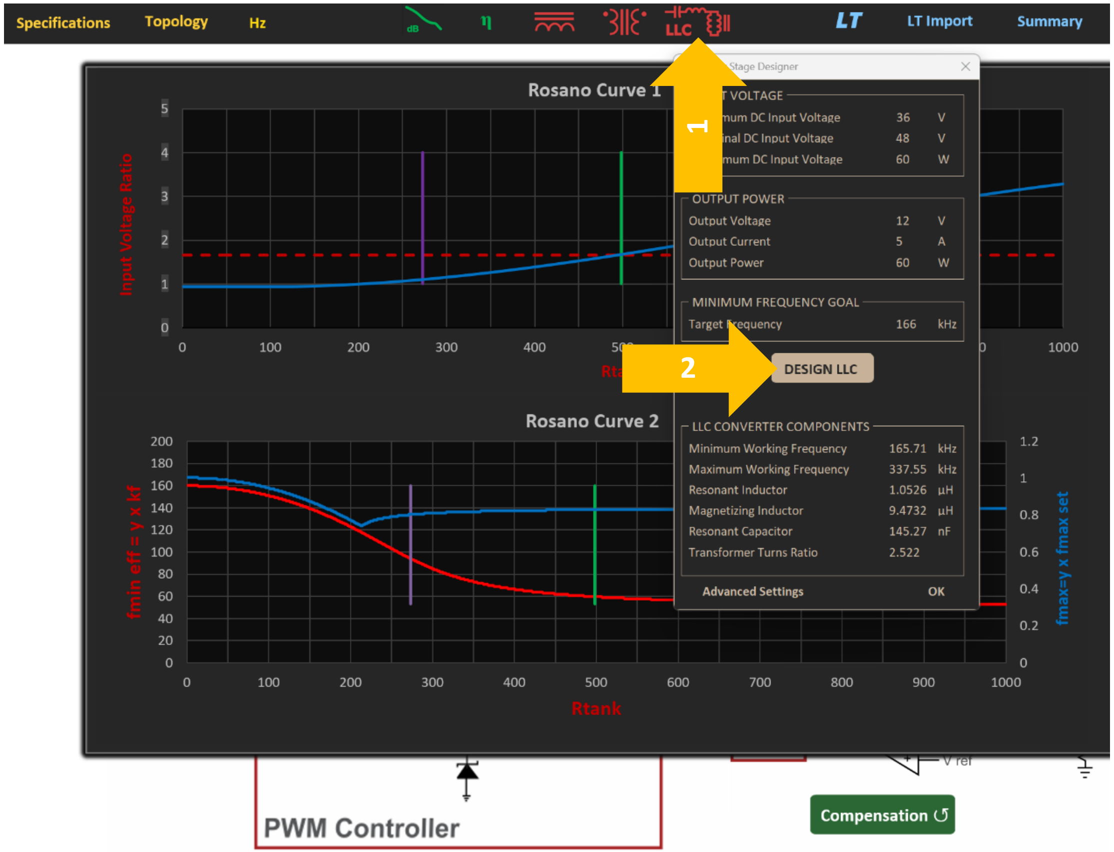


Fig. 9.6: LLC Power Stage Design Form

Once you have selected the desired LLC topology, you will see the appropriate schematic appear, along with the icon for designing the LLC circuit component values. Click on this button to pull up the LLC Power Stage Designer form.

The design process is initiated by clicking on the text highlighted with the second arrow.

Once you do this, you will see RidleyWorks® search for the optimum design point, changing the component values and frequencies at the same time.

You will then see the message that the design is complete, and simulation files with all the component values will automatically be created, ready to run in LTspice®. A total of 5 simulation files will be created and placed in a folder called C:\RidleyWorks\RidleyWorks LTspice.

LLC Steady State file must be run in LTspice before proceeding further with your design.

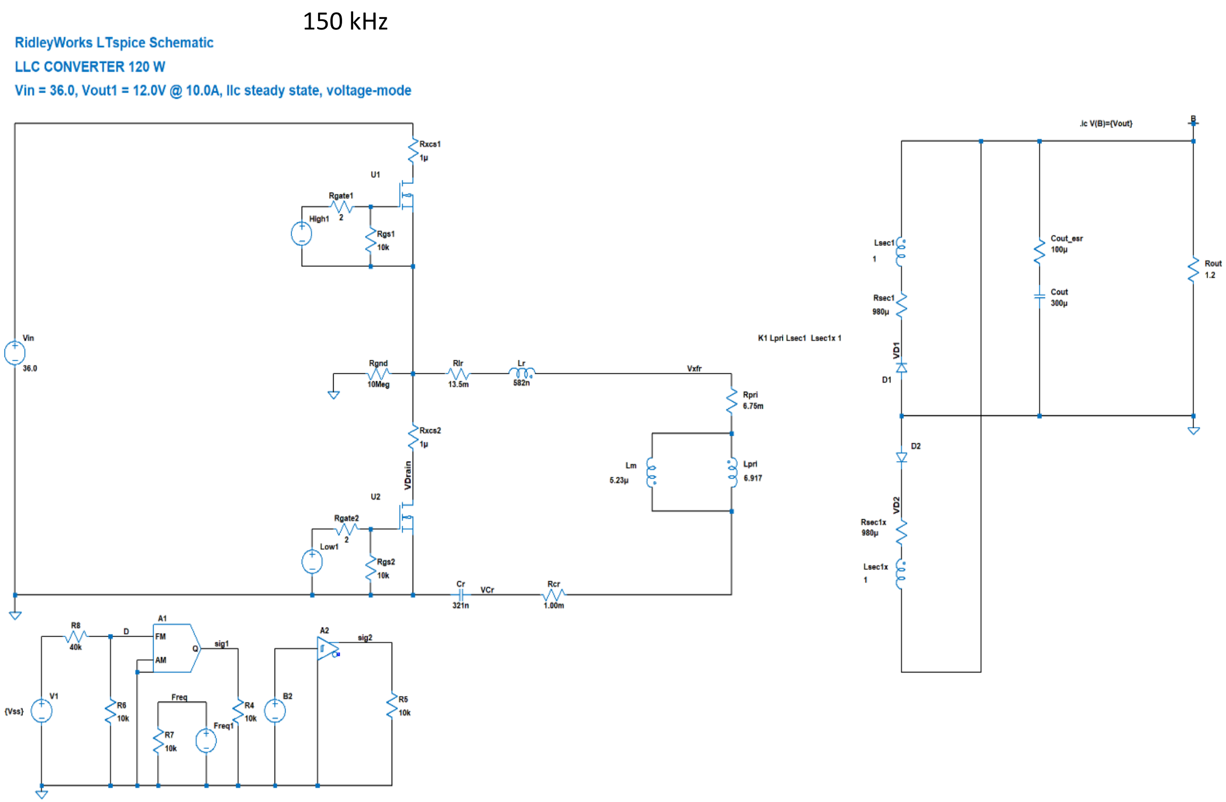


Fig. 9.7: LLC Power Stage Steady-State Simulation Circuit

9.7 LLC Circuit Simulation

Before you can view waveforms and proceed further with the design in RidleyWorks®, a simulation of the LLC converter in steady state should be run. The file created for this simulation is:

1 LLC STEADY STATE.asc

The default operating condition for the LLC design is with the minimum input voltage and full load. At this operating point, the converter should be close to the specified output voltage at the desired switching frequency selected in RidleyWorks.

If you see that the output voltage is too high, you can increase the value of **Vss**. That will increase the switching frequency and decrease the output voltage.

Conversely, If the output voltage is too low, decrease the value of **Vss** to lower the switching frequency.

You can look at the waveforms inside LTspice® to verify proper operation. Important waveforms are listed in the .save lines of LTspice®, and these can be imported into RidleyWorks®.

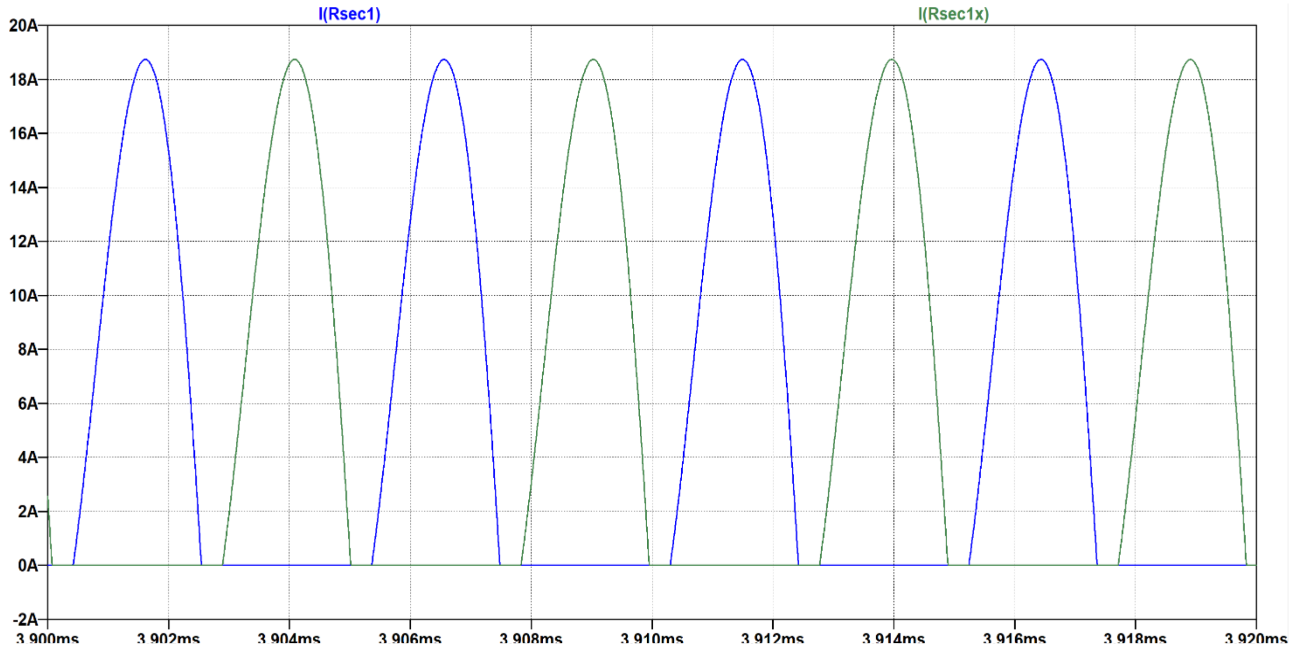


Fig. 9.8: LLC Secondary Currents at High-Line Condition. Zero -Current Switching is Maintained

9.8 High Line Operation

It is a good idea to run the steady-state simulation again with high-line input to the converter. When you do this, you will see the output voltage will be higher than the desired value. This can be reduced by raising the switching frequency of the converter by increasing **Vss**.

```
CONTROLLER PARAMETERS
.param Vss=0.761
.param FreqMax=243K
.param FreqMin=73.89K
```

The parameter Vss controls the switching frequency. A value of 5 V will set it to the maximum value, and a value of 0 V will set it to the minimum value.

When the output is regulated to the desired value, you should see that both secondary currents complete a full half-cycle and zero-current switching is maintained. This indicates proper design of the power stage components.

When you are finished verifying the waveforms in LTspice®, you can proceed with the component design in RidleyWorks®.

9.9 Exporting Waveforms from LTspice®

In the waveform window of LTspice®, click on **File** and **Export data as text.** Highlight all of the listed waveforms and click OK to save the simulation results as a .txt file. RidleyWorks® has an automated reader to import all of these values.

Note: this text file will be quite large due to the long simulation time of the steady-state circuit. You can reduce the final simulation time in the following statement if you see that steady state is reached well before the end of the simulation.

.param sim_time=8.000m

By default, the waveform data will be stored in the file named:

1 LLC STEADY STATE.TXT

Do not change this file name as RidleyWorks® will search for this file name when trying to import the data.

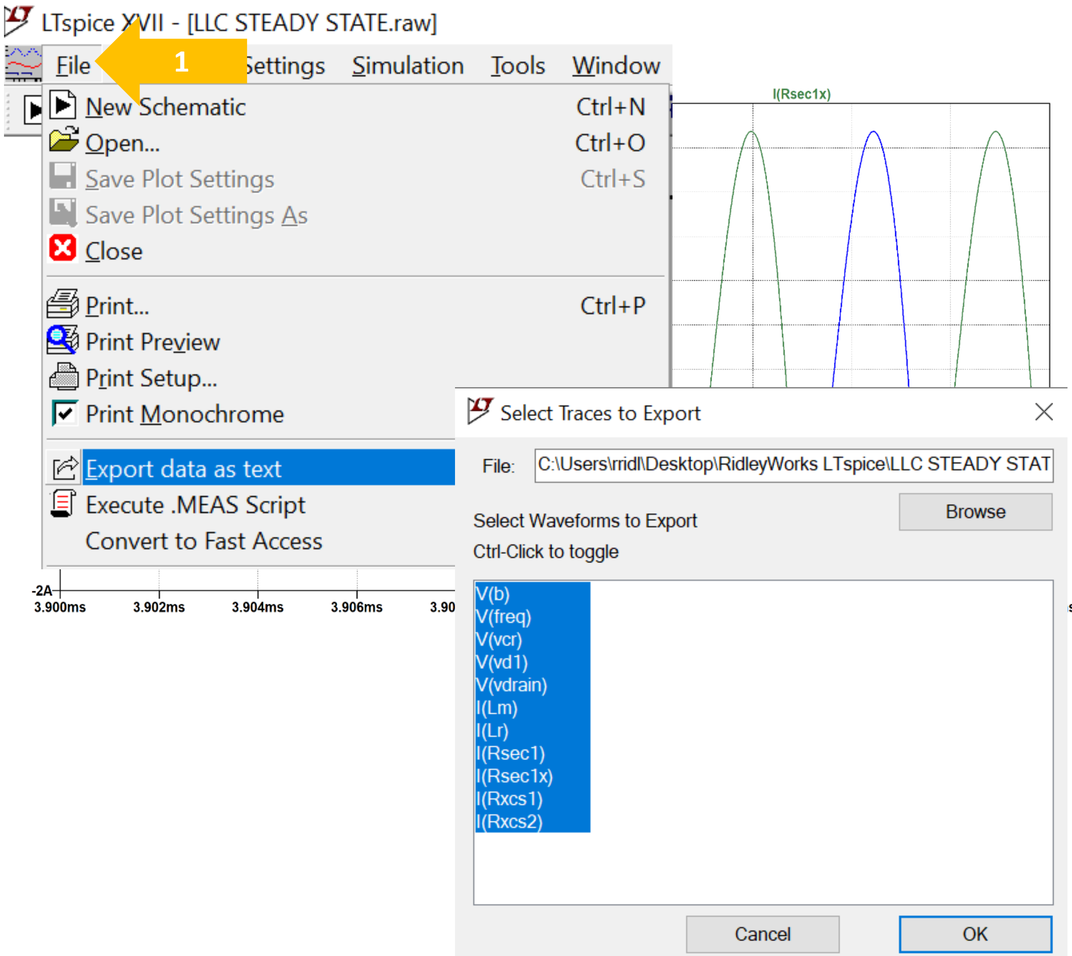


Fig. 9.9: Exporting Data from LTspice® Simulation

9.10 Importing LLC Waveforms

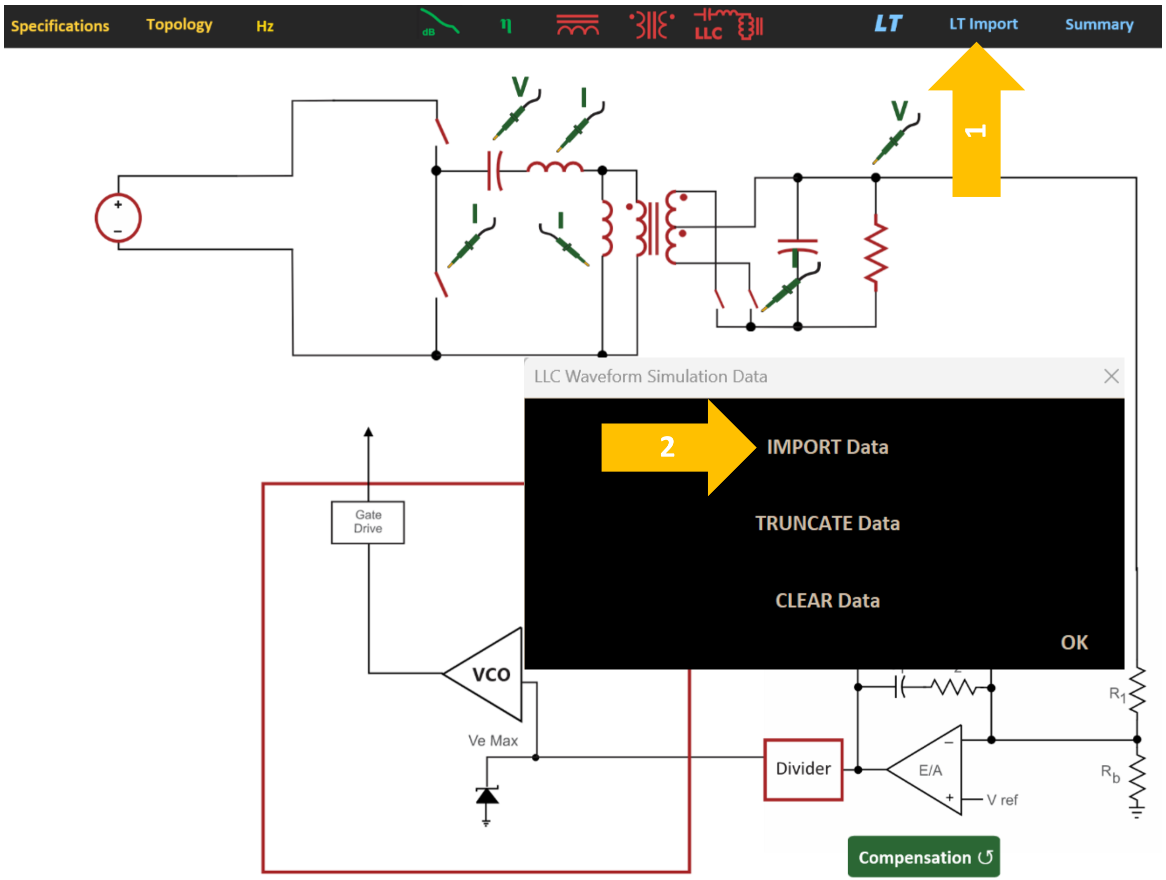


Fig. 9.10: Importing Data from LTspice® Simulation into RidleyWorks®

The waveform data that has been saved to the file 1 LLC STEADY STATE.TXT can now be imported into RidleyWorks®.

Click on the button to import the LTspice® data. You will then see a form with two options. The first option will import the large data file saved by LTspice®. You can bring in up to 1 million lines of data, but these will be truncated to the first 500 k lines. It will typically take a minute or so to import the large data set.

You also have the option to truncate the imported date to just the first 50 k lines to minimize the file size of RidleyWorks®. This step will always be automated when you exit RidleyWorks® and select the option to save the file.

The main purpose of importing the waveforms into RidleyWorks® is to be able to design the components of the power stage. In general, you will find it more useful to work on the waveform viewing inside LTspice®.

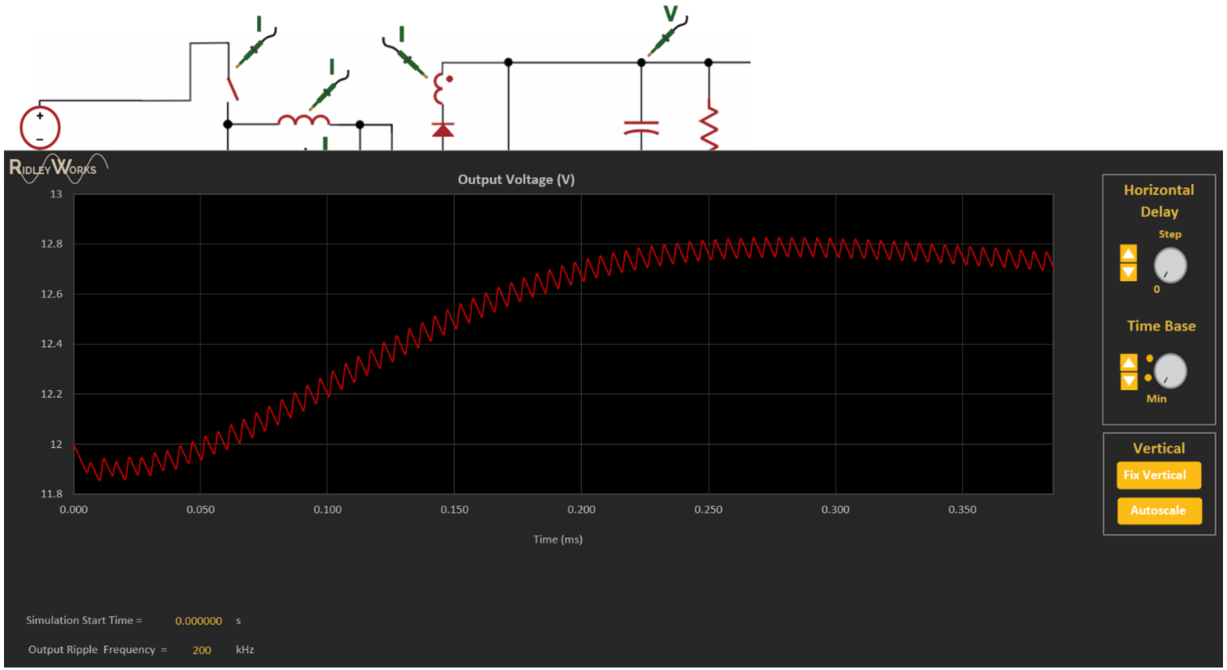


Fig. 9.11: Viewing Waveforms in RidleyWorks®

9.11 Viewing LLC Waveforms

Click on the of the green probes on the circuit schematic to see current or voltage waveforms. You can zoom in on the waveforms inside RidleyWorks® in the normal way using the timebase controls on the screen.

Once you have imported the waveforms, you can begin the transformer design. For the experienced designer, this process is straightforward, especially when assisted with the magnetics design capability of RidleyWorks®.

If you don't have magnetics design experience, you can learn more by coming to our workshops or you can work with your magnetics vendor to help you. Please note that LLC design is not familiar to many magnetics companies, and it may be better to learn the process yourself.

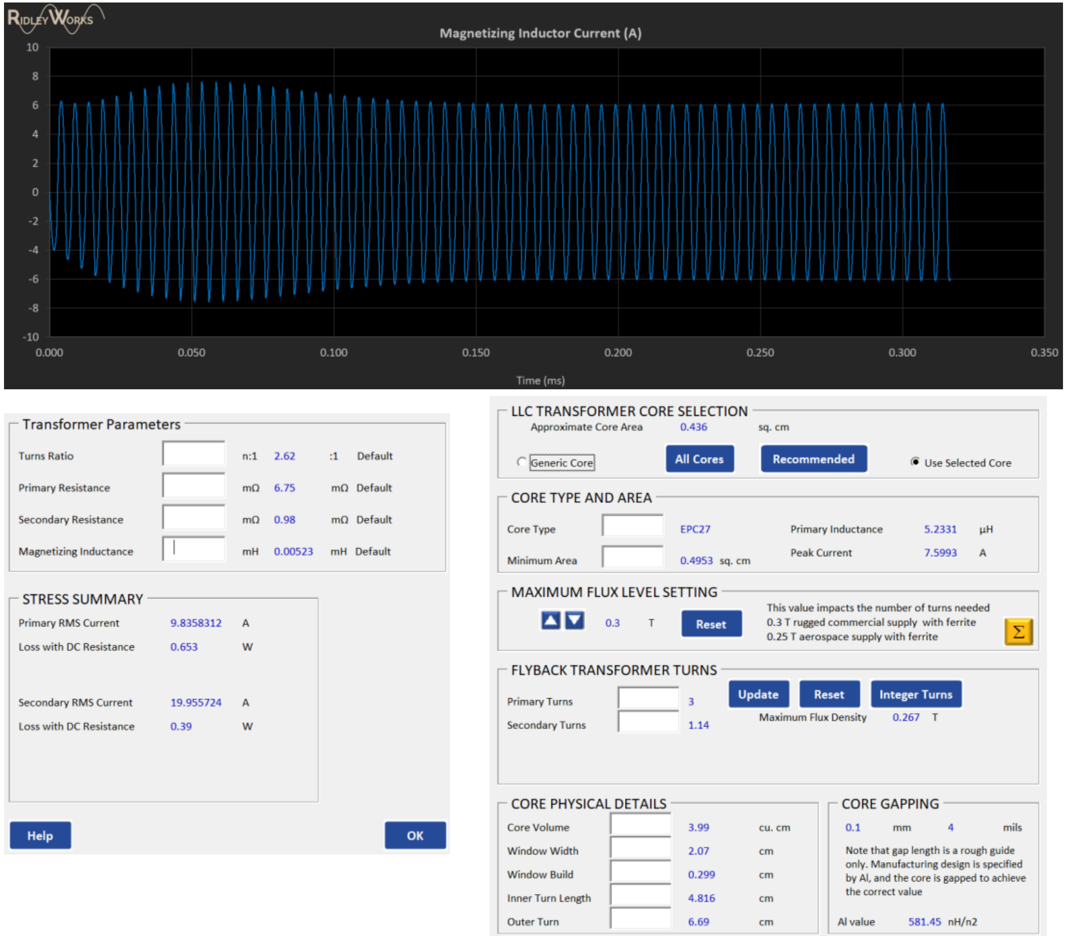


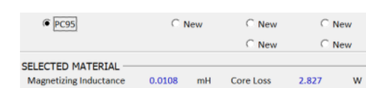
Fig. 9.12: LLC Transformer Parameters

9.12 LLC Transformer Design

RidleyWorks® treats the LLC transformer like an inductor design for the selection of the turns needed on the transformer core. It is important to simulate the converter with LTspice to see the needed energy storage in the transformer. The peak current and the magnetizing inductance value are used to make sure that the maximum flux level of the core is not exceeded.

Notice that there is an additional constraint on the turns count on the primary since there cannot be less than one turn on the secondary. The number of primary turns must be at least equal to the turns ratio of the transformer.

As frequencies are raised above 150 kHz, you may find that you increase the minimum turns count to reduce the observed core loss in the transformer. This is usually an interative design process to trade off core and copper losses. For example, with three turns on the primary, using a PC95 material, there are 3 W of core loss.



With 6 turns on the primary, the losses in the core are reduced

This is a better design choice.

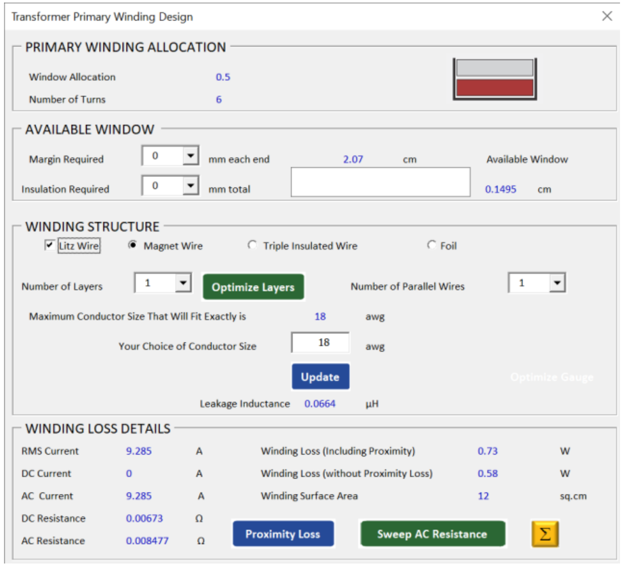


Fig. 9.13a: LLC Transformer Primary

9.13 LLC Transformer Winding Design

Once the turns count is established for a specific core and turns ratio, we turn our attention to the detailed winding layout in the transformer. For this design, six turns are used on the primary of the transformer. If you try to use solid wire, the proximity losses are significant due to the high ac content of the primary transformer waveform. Switching to Litz wire reduces the losses substantially.

As with standard transformers, RidleyWorks estimates the loss of the winding including proximity effects. For experienced designers, you can go deeper and ask RidleyWorks to sweep the AC resistance versus frequency in order to build an LTspice model. This step is extremely powerful when estimating the losses in the transformer at different operating points.

You could also choose to use a foil for the primary. The disadvantage of this approach is that it will increase the capacitance of the transformer and affect the operation of the LLC converter. As with any design, it is crucial to measure the actual performance on prototype hardware (and later on production hardware) to ensure that the parameters of the transformer are close to the specified values. Engineers with either the AP300 or the RidleyBox will be aware of how to easily do this.

After you have designed both the primary and secondary windings, you can use the leakage inductance estimation of the primary winding to begin the design of the resonant inductor of the LLC.

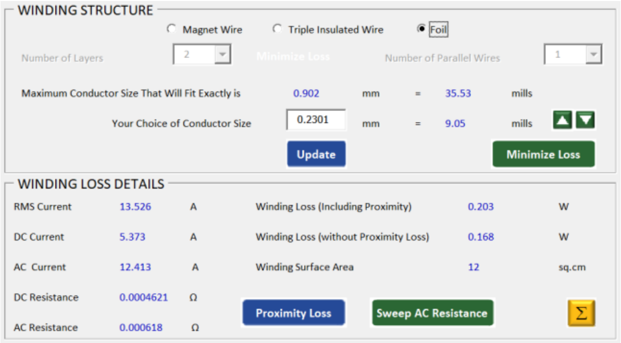


Fig. 9.13b: LLC Transformer Secondary

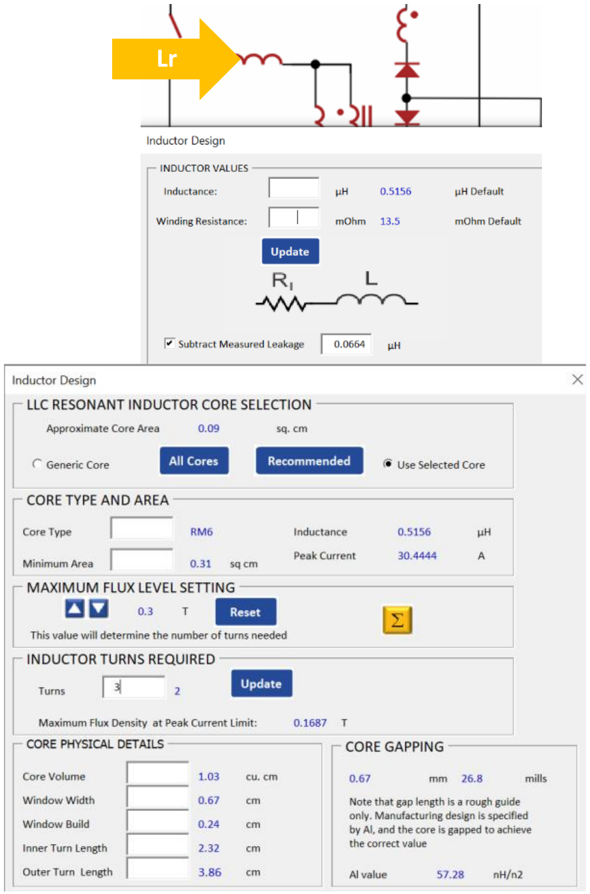


Fig. 9.14: Resonant Inductor Core Selection

9.14 LLC Inductor Core and Turns Design

The first step in designing the LLC series inductor is to determine the value of the discrete inductor to be added. The transformer leakage inductance will provide a portion of the resonant inductor value. You can account for this contribution by clicking on the schematic of the converter. Enter the value of the leakage inductance (make sure you measure this with your AP300 or RidleyBox to confirm the value at the switching frequency) on the primary side of the transformer. You will see the required amount of extra inductance needed changing as this leakage number is changed.

Some LLC transformers are deliberately designed with very high leakage so that an additional inductance is not needed. In our experience with industry LLC converters, the high-efficiency products add a discrete external inductor in most cases.

As with the transformer design, you may see that the minimum number of turns suggested (2 in this case) results in high core loss. Increasing the number of turns to 3 will produce a more optimal design.

9.15 LLC Inductor Winding Design

For this design, we only need three turns on the inductor. The losses are minimized by using a foil structure with a 15-mil thickness foil. Notice that it is possible to use a thicker foil, but the high AC currents would result in higher proximity losses in the winding.

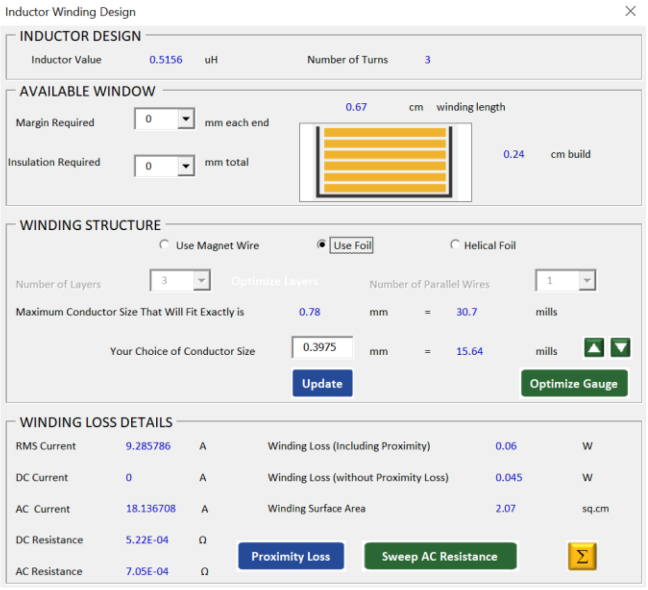
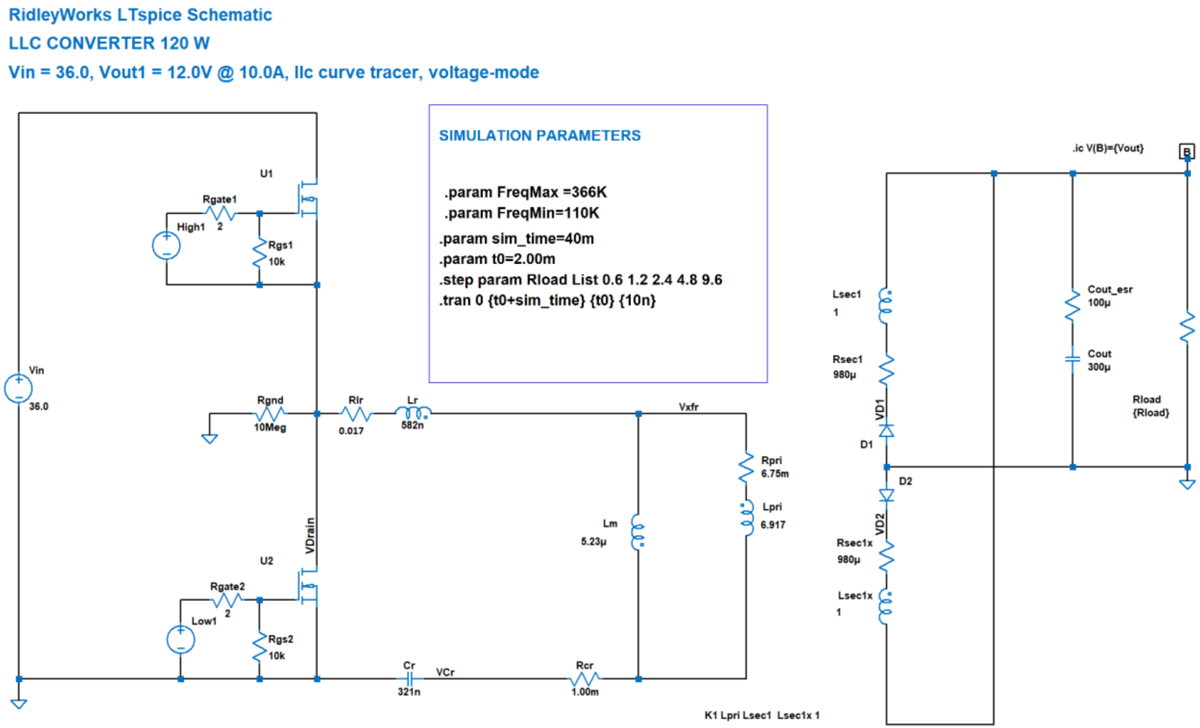


Fig. 9.15: Resonant Inductor Winding



9.16 LLC Gain Curves Schematic

RidleyWorks® generates a second LTspice simulation file for the LLC converter named:

2 LLC CURVE TRACER.asc

When you run this file in LTspice, the frequency will be increased from the minimum value to the maximum value over the duration of the simulation.

.param FreqMax =366K
.param FreqMin=110K

The minimum and maximum values from RidleyWorks should cover the entire range of operation of the converter from high line down to low line.

Please note that the accuracy of the curves will be impacted by the total simulation time. If the time is too short, there will be portions of the curve that have sharp changes in them. If you increase the time, and the curves do not change, then you can be confident the curves are accurate. In this example, the default simulation time of about 6 ms is increased to 40 ms for optimal results.

.param sim_time=40m

Fig. 9.16: Schematic for Generating LLC Gain Curves

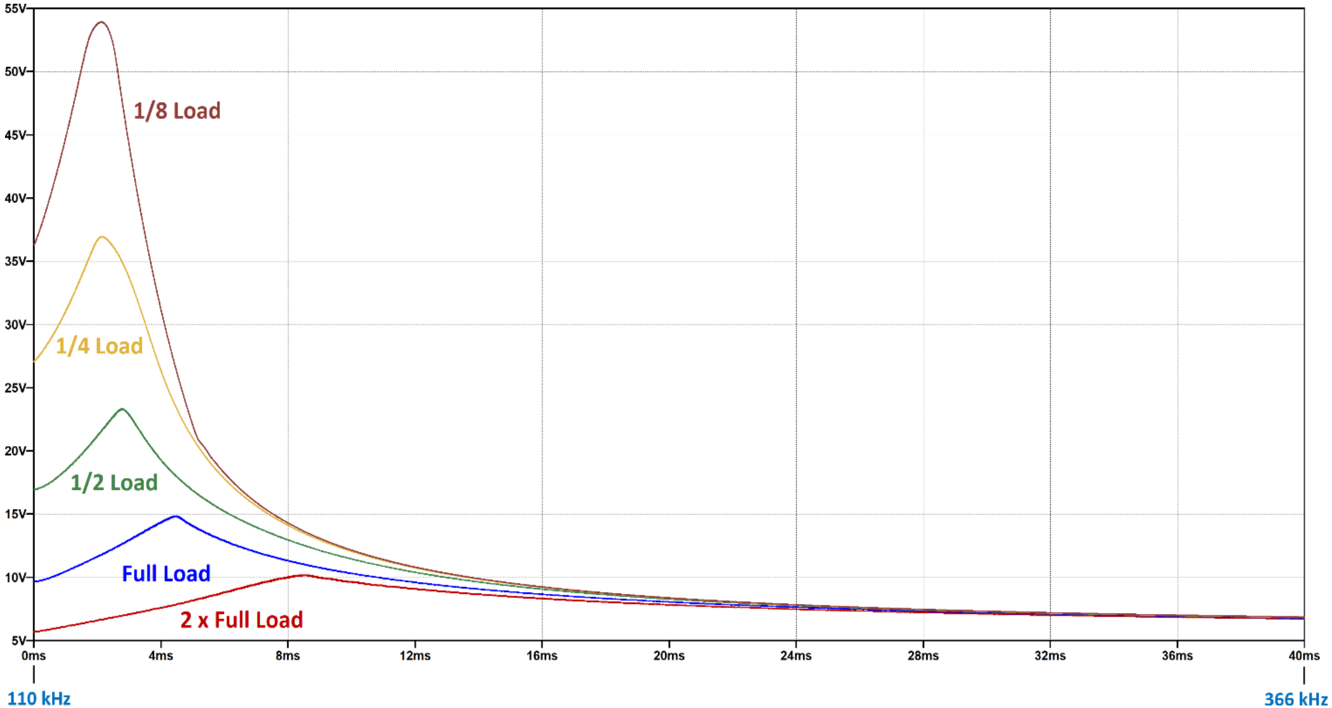


Fig. 9.17: LLC Gain Curves for a 12-V 10-A Output with a 36-V Input

9.17 LLC Gain Curves

The LLC gain curves show the output voltage for your specific converter design at different load levels. For the lowest curve in red, the load is set at twice the specified value. In this case, output voltage cannot reach the specified value of 12 V and the converter cannot regulate.

At full load, the curve shown in blue shows that the output voltage reaches the desired regulation value just above the resonant peak.

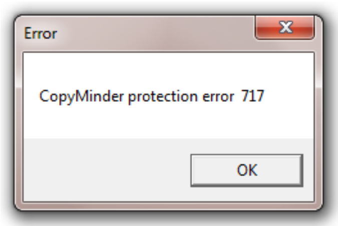
The curves generated are the actual curves including all the losses of your particular circuit. This is much more useful than working with theoretical curves created by first harmonic analysis methods.

You can spend a lot of time working with these curves to understand how your converter is working. For example, you will want to redraw them for high-line input to determine the actual limit to the switching frequency needed.

Appendix RidleyWorks®

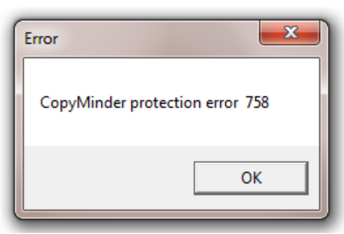
Appendix A Error Messages

Protection Error 717 Incorrect Passkey Entered



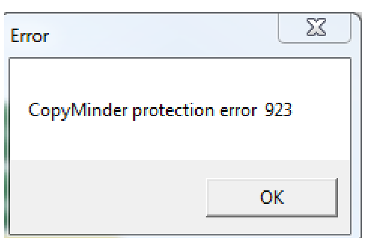
You will see this message if you enter an invalid passkey for the program. Check for the proper number, and contact Ridley Engineering if you cannot find it.

Protection Error 758 License Expired



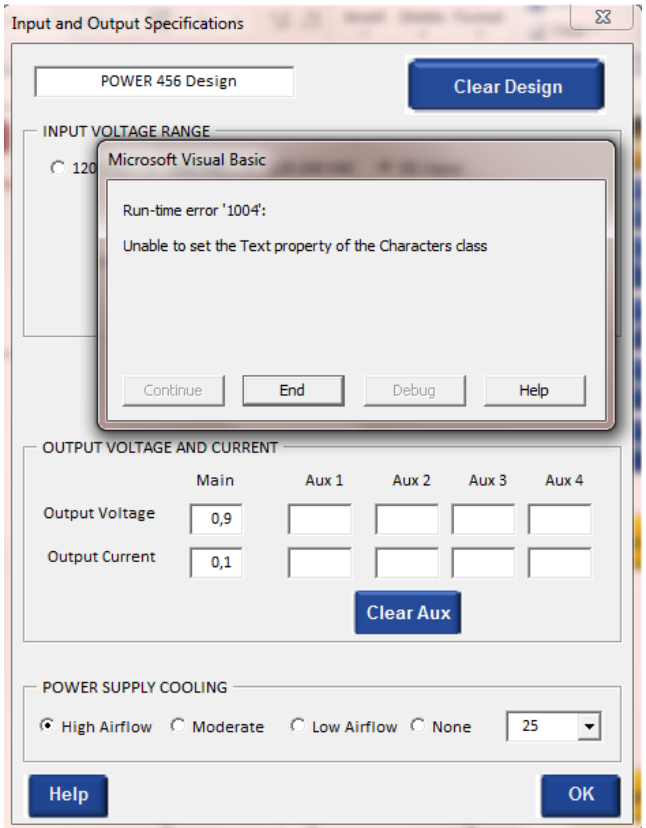
You will receive this error when your license has reached its expiration date. Contact us to verify the date or to renew your registration.

Protection Error 923   Installation Incomplete



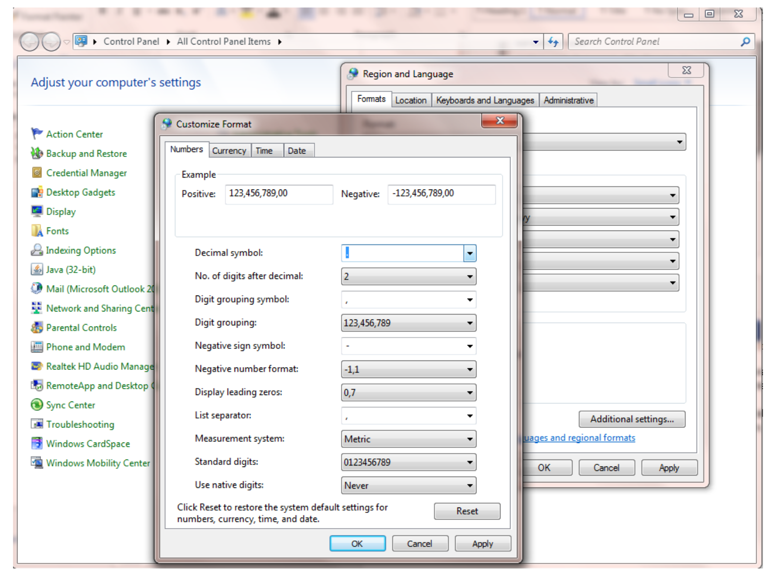
You may receive this error if the software installation did not finish generating all of the necessary files. Please go to Section 1.4, item number 2 of this manual to learn how to complete the installation.

Run-Time Error 1004



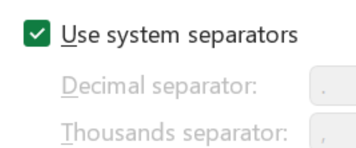
This error will occur if you have an old version of Excel prior to 2013. Please upgrade to the latest version.

In some cases, the error will occur if you do not have the decimal separator character set to a period “.” You can change the setting of the decimal separator in your Windows system as described below.

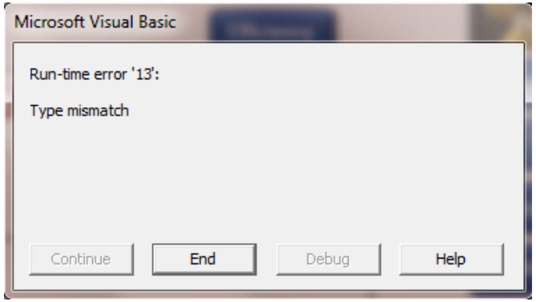


In the windows search bar pull up the Control Panel, Region and Language. Under the Format tab, click on Additional Settings, and you will be able to change the character to a period as shown.

Also make sure that Excel is using the system separators. Go to File Options Advanced in Excel and check the box shown here:

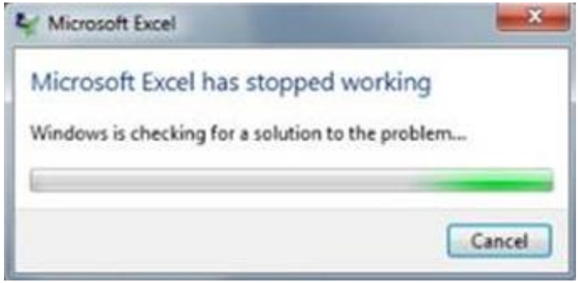


Data Entry Format Error – Run-Time Error 13



Most mistakes in data entry will be flagged by RidleyWorks® and will give you an error message. However, if you see the message here, you may have the decimal separator set incorrectly in your system. This is changed as described in the above section. Or, you may have used some other invalid numerical character. If you see this happen, please try to describe the sequence of events to us or send us the file. We are always working to improve and update the program.

Microsoft Excel Stopped Working (EMET problem)



If you get the message that Excel has stopped working while verifying the license, it may be because your IT department has installed Microsoft EMET (Enhanced Mitigation Experience Toolkit). This will block access to the required website to verify the license code.

It is necessary to disable the following EMET options for RidleyWorks® to register itself properly without triggering EMET to shut down the program.

Export Address Table Filtering
Simulate the execution flow

There is no security risk in implementing these changes.

Object Doesn’t Support Property or Method

You may see messages such as this when you click on one of the buttons. Microsoft did some security updates recently, and it causes problems in excel. You can find it discussed at:

https://social.technet.microsoft.com/Forums/exchange/en-US/3f29c84b-97ad-46f7-9bc0-8648d69336ce/kb2553154-breaks-our-excel-macros?forum=officeitproprevious

fix is to simply delete the file MSForms.exd from any Temp subfolder in the user's profile. For instance:

```
C:\Users\[user.name]\AppData\Local\Temp\Excel8.0\MSForms.exd
C:\Users\[user.name]\AppData\Local\Temp\VBE\MSForms.exd
C:\Users\[user.name]\AppData\Local\Temp\Word8.0\MSForms.exd
```

You can search for any files ending in .exd and delete them while excel is closed. Then start excel again and it should work properly. Microsoft are aware of this issue and are trying to fix it, apparently.

More information is at http://stackoverflow.com/questions/27411399/microsoft-excel-activex-controls-disabled

LTSpice Cannot Find Sub or Sym Files

On installation, the special symbols and subcircuits used by RidleyWorks to simulate fast and efficiency are placed in the

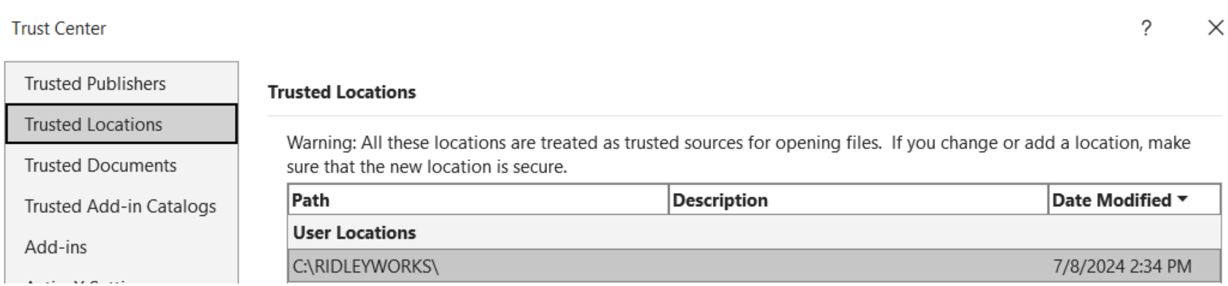
C:\RidleyWorks\LTspice folder by default.

The folders are also placed in the directory:

C:\Users\<username>\AppDate\Local\LTspicelib folders, consistent with the current installation of LTspice (April 2024).

Enabling Macros in Excel

In Excel's Trust Center under File > More > Options >Trust Center
Add the C:\RIDLEYWORKS folder to allow Excel to always trust running macros when RidleyWorks is launched in Excel.



Copyminder Error 1991

IT teams become more and more cautious as time goes on. The protections at your company may be stopping files from running. Copyminder in the UK has provided this information:

This error means the 32 bit helper process for the 64 bit protection check that our code has just written out to the disk without an error being triggered could not be started. We attempt to start it through the standard Windows kernel API and this returns a failure. The file in question is the one that ends in .cm64.exe and written out in the same directory as the license file.

Here is the solution from a RidleyWorks user:

Just to follow up, this was being flagged by a program our company uses called SentinelOne Agent. Our IT department was able to make a change to permit this file, and now it works fine.

Please show your IT this information and they should be able to fix the issue.

Appendix B License Agreement

Ridley Engineering, Inc.
Software License and Limited Warranty Statement

SOFTWARE LICENSE

1. License Grant. Ridley Engineering, Inc. or Ridley Engineering Europe UK Ltd. (referred to as "Ridley") grants to the registered purchaser (referred to as "you" and "your") a nonexclusive right to use one copy of the enclosed computer programs ("the PROGRAMS") on a single computer in accordance with the terms of this Software License. Ridley reserves all rights not expressly granted to you in this software license. This license is effective until terminated. Ridley may terminate this license if you fail to comply with this Software License. In the event of such termination, you agree to destroy the original and all copies of the PROGRAMS.

2. Copyright. The PROGRAMS are licensed, not sold, to you. Ownership of the PROGRAMS, and their associated proprietary rights are retained by Ridley and its assignees. The PROGRAMS are protected by the copyright laws of the United States and other countries, and by international treaties. Therefore you must comply with such laws and treaties in your use of the PROGRAMS and, among other things, you may not copy the PROGRAMS except that you may load them into the random access memory of your computer, install the PROGRAMS on a hard disk or other storage device of a single computer, copy versions of the PROGRAMS as you modify their data contents onto a single computer, and make a single copy for archival purposes. Any copies of the PROGRAMS, either in their original forms, or modified by you, are subject to this Software License. You may not copy the written material accompanying the PROGRAMS without the prior written consent of Ridley.

3. Transfer and Use Restrictions. You may not sell, lend, or otherwise transfer or lease the PROGRAMS or this license to others without written permission from Ridley

LIMITED WARRANTY STATEMENT

90-Day Limited Warranty. Ridley warrants that the PROGRAMS will perform substantially as provided in the accompanying written materials for a period of ninety (90) days from the date of receipt from Ridley.

This Limited Warranty gives you specific legal rights, and you may have other rights which vary from state to state and from country to country.

Remedies. Ridley's entire liability, and your exclusive remedy for any failure of the PROGRAMS to comply with the Limited Warranty shall be, at Ridley's option: (a) repair or replacement of the PROGRAMS, or (b) return of the price you paid for the PROGRAMS. You must return all copies of the PROGRAMS and written material to Ridley or Ridley's dealer with a written explanation of the noncompliance and a copy of your receipt within 90 days of the date you received the PROGRAMS to receive this remedy. Any replacement PROGRAMS will be warranted for the remainder of the original 90-day warranty period, or 30 days from the date you received the replacements, whichever is longer.

DISCLAIMER OF WARRANTIES. Ridley does not warrant that the programs are free from all bugs, errors, and omissions. Except for the 90-Day Limited Warranty described above, Ridley makes no other warranties, express or implied, with respect to the PROGRAMS and any accompanying material; RIDLEY DISCLAIMS ANY IMPLIED WARRANTY OF MERCHANTABILITY, AND RIDLEY DISCLAIMS ANY IMPLIED WARRANTY OF FITNESS.

LIMITATION OF LIABILITY

DAMAGES LIMITATION. Except for the remedies set forth in the Limited Warranty statement, Ridley shall have no liability or obligation for any other damages or remedies, including, without limitation, the cost of substitute goods and any direct, incidental, special or consequential damages, arising out of this Software License and Limited Warranty statement or the use or inability to use the PROGRAMS. Some states do not allow the exclusion or limitation of incidental or consequential damages, so the above limitation may not apply to you.

LIMITATION OF LIABILITY. In no event shall Ridley's liability (whether in contract, warranty, tort (including negligence), product liability or other theory) with regard to the PROGRAMS, the accompanying written materials, or this Software License and Limited Warranty Statement exceed the compensation you paid to Ridley for the PROGRAMS.

If you wish to contact Ridley Engineering for any reason, including warranty service, please write to: Ridley Engineering, Inc., Ridley Engineering, Inc., 601 E. Daily Drive, Suite 112, Camarillo, CA 93010

EMail: info@ridleyengineering.com