ST WDS Installation Guide, User Guide and Design Example

Biricha Digital Power Ltd

Table of Contents

Table of Contents 2

Downloading and Installing ST WDS 3

Using WDS 7

Design Example 1: Voltage Mode Buck Converter 8

Design Example 2: Digital Voltage Mode Buck 15

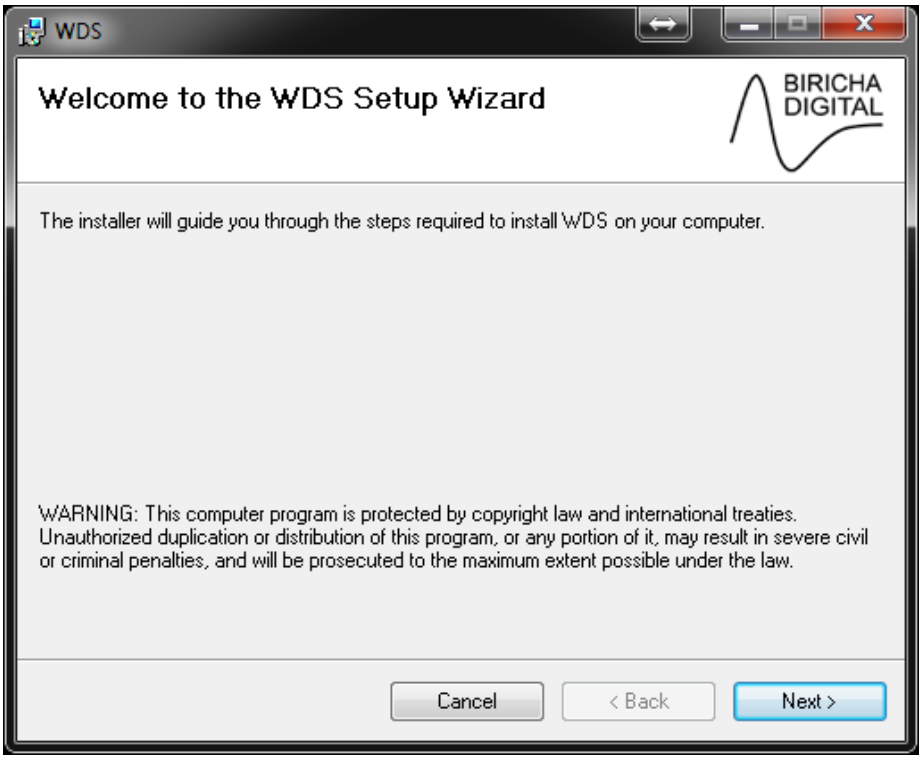
Other Options 20

Downloading and Installing ST WDS

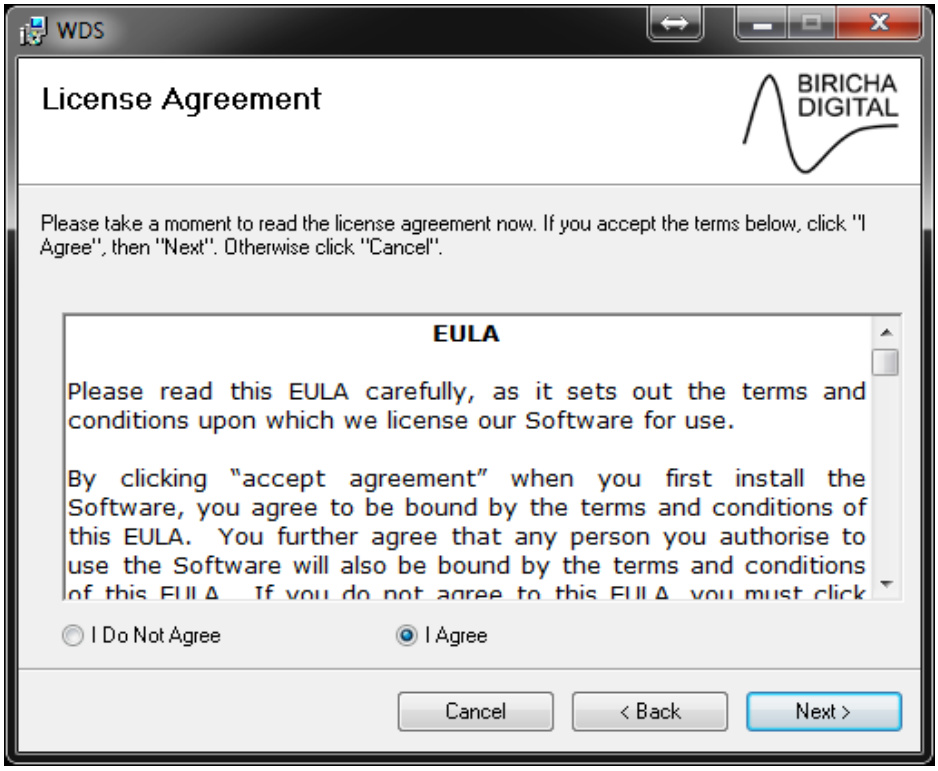
First, please download the latest version of ST WDS from www.biricha.com/st-wds

After downloading the WDS setup files, run the setup to being the installation.
You may receive a warning from Windows - click "More Info" and “Run Anyway” to proceed.

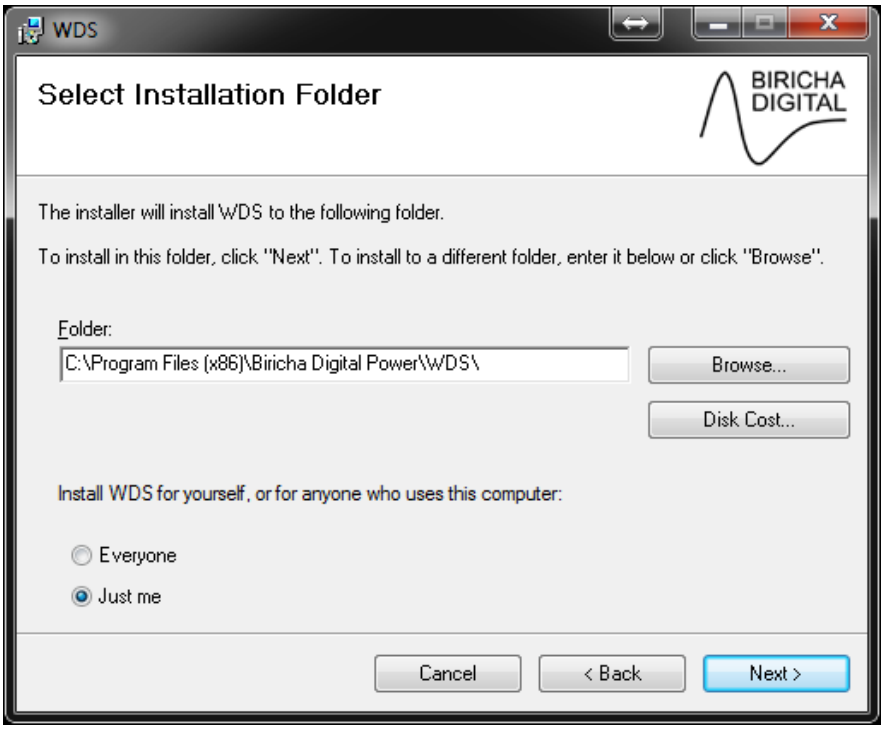
When the installation beings you will be presented with the “Welcome” screen. Click "Next".



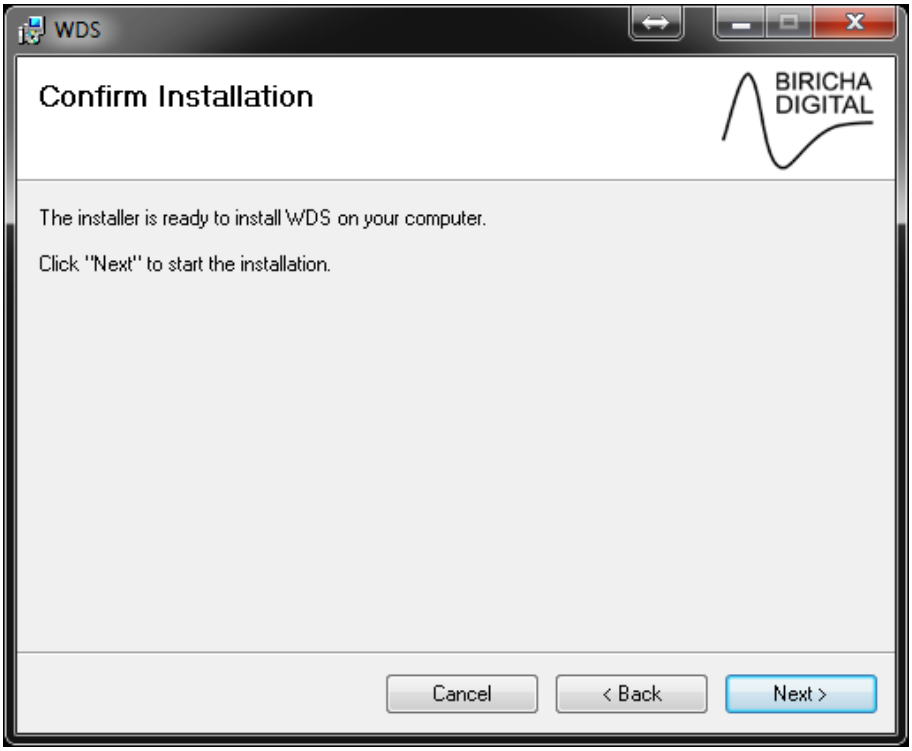
Please read the end-user license agreement (EULA) carefully. If you agree with the terms, select "I Agree" and click "Next" to continue. If you do not agree with the terms of the EULA then you should not proceed with the installation.



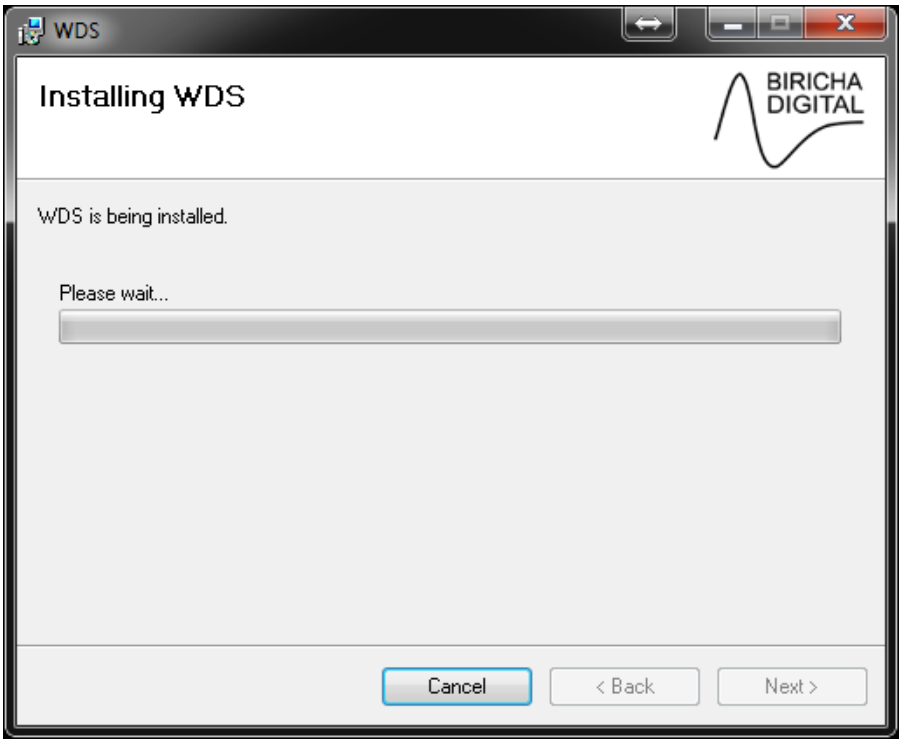
Select the installation folder for WDS. The default installation folder is your "Program Files" directory. Click "Next" to continue.



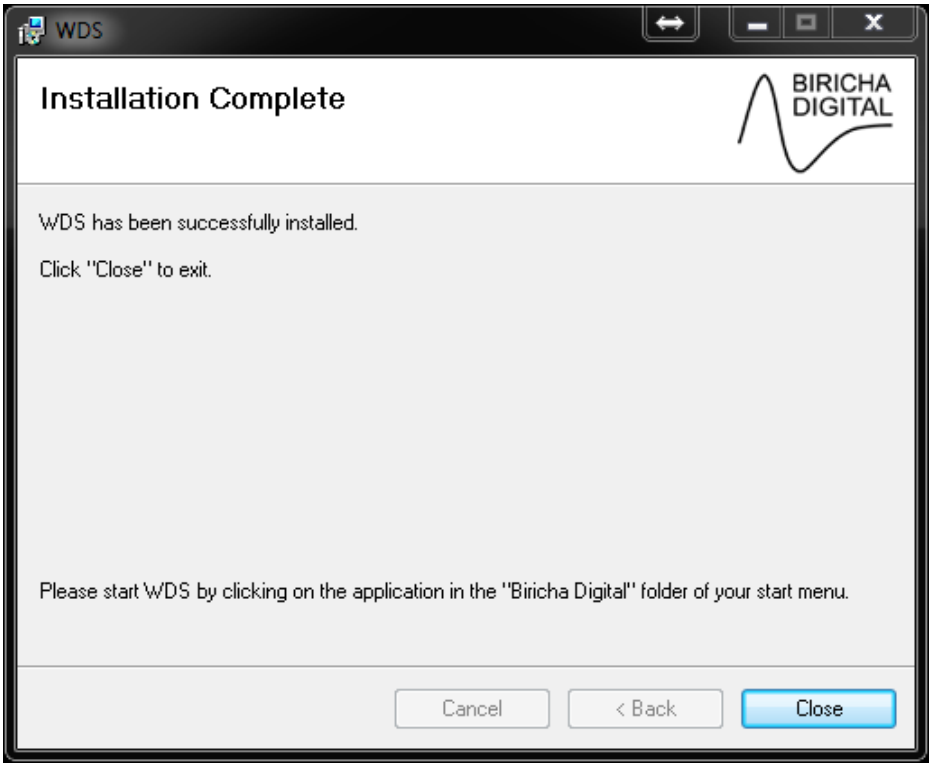
Click "Next" to proceed with the installation.



You may receive a message about User Account Control from Windows, if so, click "Yes" to proceed with the installation.



The installation is now complete. Click "Close" to exit the installer.



Setting up Licensing

When downloading ST WDS you will have been presented with an activation code. Please copy this from your web browser to your clipboard. Then got to Help -> License and then paste your activation code into the relevant box. This will activate WDS.

If you do not have an activation code then WDS will run in trial mode. Under trial mode you have full access to all functions of WDS; the only limitation is that the input and output voltages are locked.

Updating WDS

In future, if your license file permits, WDS will prompt you when updates are available. Please ensure that you always update to the latest version.

Using WDS

Concept

- To begin, the user should select their preferred topology and enter an input voltage, output voltage and output current specification
- WDS will then calculate all other parameters based on this
- The user will then select parts from their supplier and then enter the actual values into WDS
- The design should start on the "Specification" tab and work through the tabs systematically. i.e. "Specification" to "Transformer" to "Semiconductor" to "Output Filter" to "Controller Design" etc.
- The ideal calculated values will automatically update as the users enters the actual values (such as selected transformer turns ratio, output capacitor, inductor, etc.)

Combo Boxes

- Combo boxes, such as the one shown below, are used to switch between WDS calculated values and user entered values
- WDS calculated values are shown with a blue background:

- If the user overwrites this with their own value, the background colour changes to white:

- WDS will use whatever value is currently active in the combo box. It is important to note that WDS will always use the user entered value over the WDS calculated value so if the user ever overwrites an calculated value, WDS will give priority to the user entered value.
- For your convenience, the nearest preferred values are calculated for many components and these can be selected by clicking the arrow to the right of the combo box
- The user can always switch back to the default WDS calculated value by clicking the arrow to the right of the combo box and selecting the value next to the text "(default)"

Calculated Values

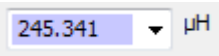
- The locked text boxes on the left hand column indicated the original WDS calculated value and allow the user to compare the calculated value to the user defined value (if entered)

Design Example 1: Voltage Mode Buck Converter

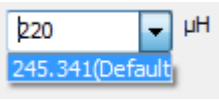
Preamble

Within WDS you will be presented with a selection of tabs, edit boxes and combo boxes. There is a general design methodology is that you should start with the "Specification" tab and work your way across to the "Analog (Non-Isolated)" tab updating the relevant parameters with your design choice along the way.

The default calculated values are always highlighted in purple as shown in the picture below.

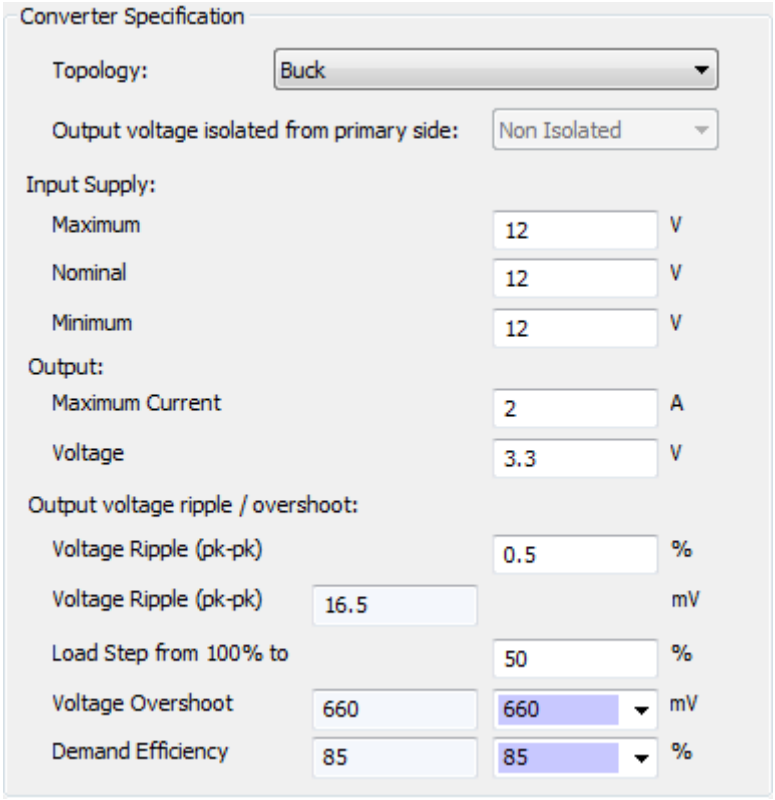


You can edit this at any time by clicking into the box and overwriting them with your own value. The box will no longer be highlighted in purple indicating that your own value is being used. The value typed in by the user will always take precedence over the WDS calculated value. If at any time you wish to return to the default calculated value you can do so by clicking on the drop down arrow on the right hand side of the box and then selecting the option that says "(default)" next to it as shown in the screenshot below.



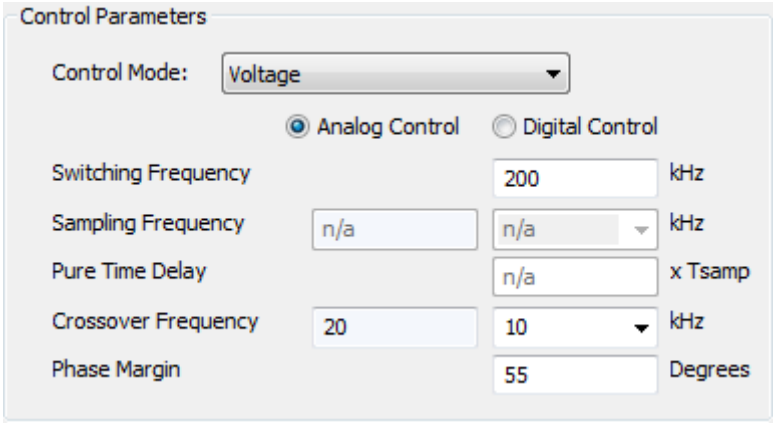
Specification Tab

On the "Specification" tab, enter the following specification into the relevant fields under "Converter Specification":



As you update the values, WDS will automatically update the design and show the associated Bode plot in the Frequency Response tab.

Again on the "Specification" tab, enter the following specification into the relevant fields under "Control Parameters":



Note: The “Analog Control” option is not available within ST WDS. This feature is only available in the full version of WDS available form www.biricha.com/wds

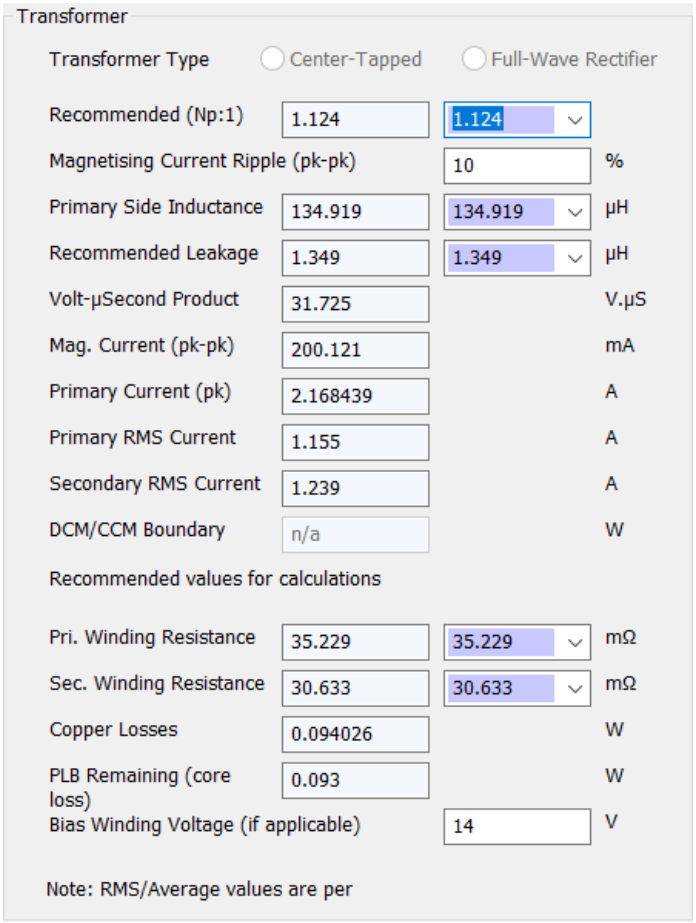
Transformer Tab

Of course, the transformer tab is not applicable to the Buck converter. The boxes within this tab will be greyed out for all non-isolated topologies. However, for completeness we describe them here.

For an isolated power supply you would see a tab as shown below.

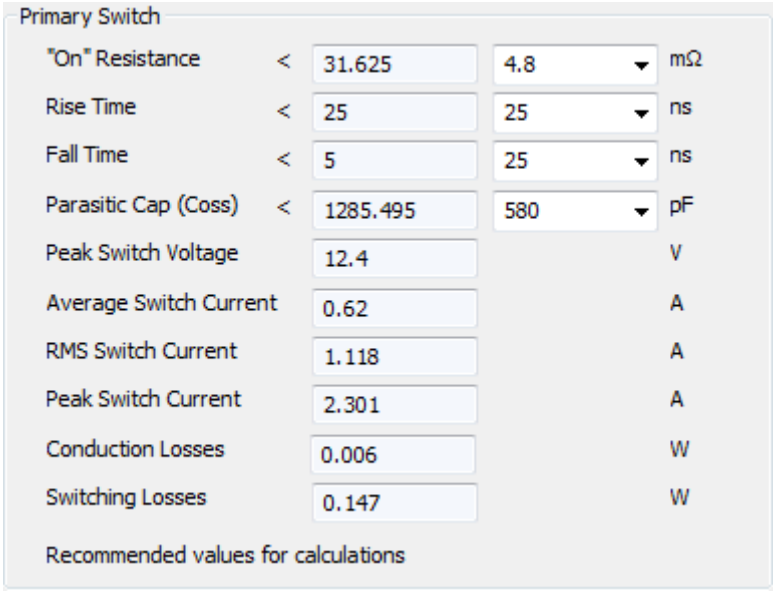
Recommended (Np:1) gives the turns ratio of the transformer. The Primary Side Inductance of this transformer is calculated based on the user defined peak to peak ripple of the magnetizing current (Magnetizing Current Ripple (pk-pk) and the Secondary Side Inductance can then be calculated from the turns ratio. Finally recommended leakage is simply a set percentage of the Primary Side Inductance.

All worse case stresses are then calculate; e.g. Volts-Microseconds product. Note that Bias Winding Voltage is for documentation only and is not used.



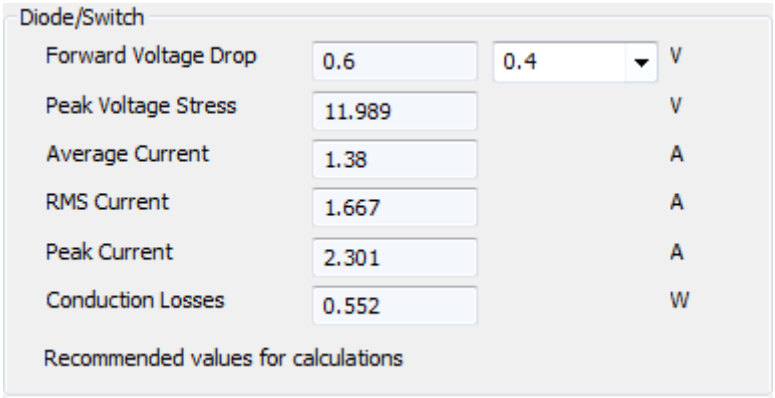
Semiconductors Tab

Now move onto the "Semiconductors" tab, enter the following specification into the relevant fields under "Primary Switch"; these are the actual parameters of the FETs that we are using on the board. Important: for now, please ignore the values in the left hand column; these are calculated values based on your specification and will get updated as you input various parameters in the future tabs.



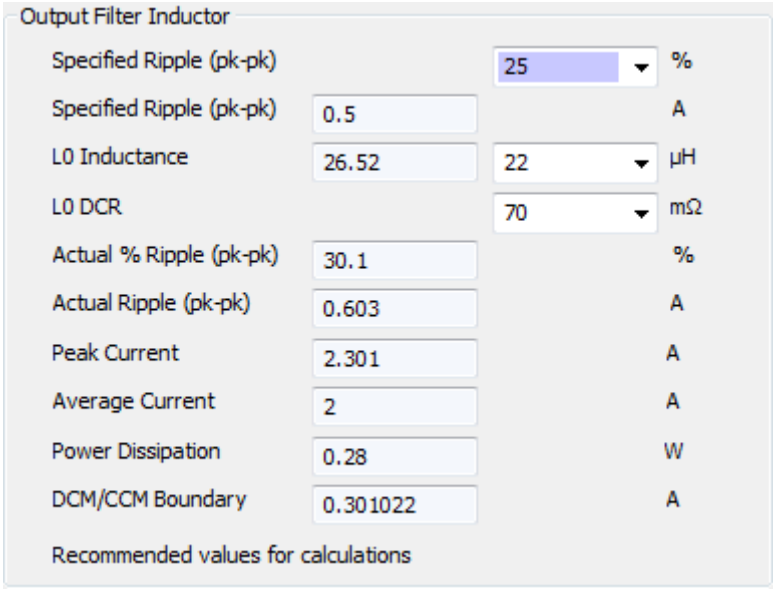
These are the values for the semiconductor switches that we are using on the Biricha Buck board used in our workshops.

Again on the "Semiconductors" tab, enter the following specification into the relevant fields under "Diode/Switch":

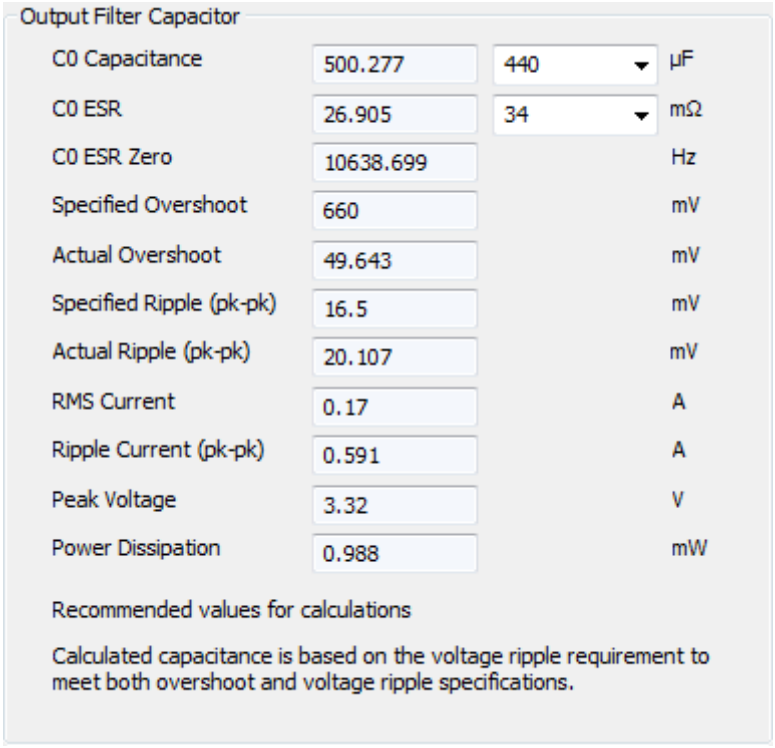


Choke/Output Capacitor Tab

Now click on the "Output Filter" tab. On the "Output Filter" tab, enter the following specification into the relevant fields under "Output Filter Inductor ":



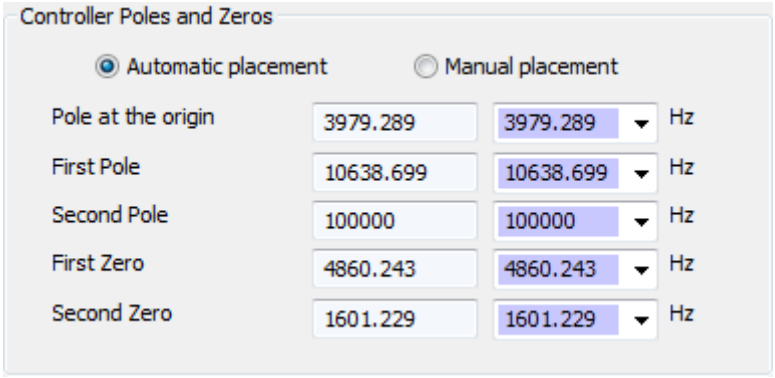
Again this is the parameters of the Wurth inductor that we are using on the Biricha Buck board. Also on the "Output Filter" tab, enter the following specification into the relevant fields under "Output Filter Capacitor ":



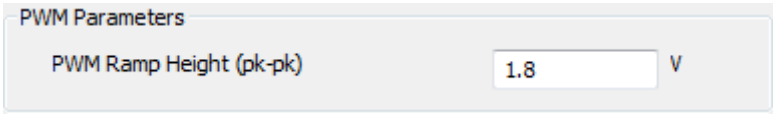
Note that we are using 2x220uF caps and we measured the ESR to be around 70mΩ for 1 capacitor. So the ESR of 2 in parallel will be around 34.5mΩ.

Controller Design Tab

On the "Controller Design" tab, WDS has selected a Type III compensator. The automatic pole/zero placement algorithms have calculated the location of the poles and zeros such that the loop will achieve the crossover frequency and phase margin specification concurrently, i.e. WDS has designed a compensator that will give a crossover of 10kHz and a phase margin of 55 degrees. As shown below:



Note that in this tab WDS asks for the PWM ramp height which has been set to 1.8V



Note that this value is chip dependent. For the analog chip used in this example the typical value from the datasheet is 1.8V.

If not already in context, click on the "Frequency Response" tab on the right hand side to view the simulated Bode plot. Under the “Bode Plot” section of this tab, make sure that only “Loop” is ticked. This will display the simulated loop response. See that the simulated crossover frequency and phase margin are displayed at the top of this tab.

Analog (Non-Isolated)

Note: The “Analog (Non-Isolated)” option is not available within ST WDS. This feature is only available in the full version of WDS available form www.biricha.com/wds

Click on the "Analog (Non-Isolated)" tab. The component values required for the Type III compensator have been automatically calculated.

In this example we will be using an analog control IC to implement the controller.

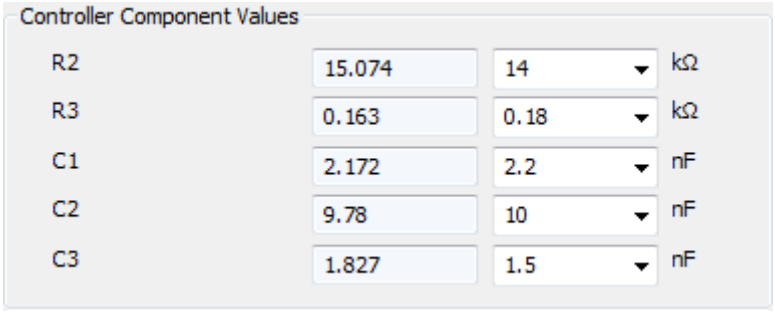
The voltage reference from this chip is 5.1V. But, in this example, we divide this by 2 using a potential divider. Therefore in WDS please make sure that the “Error Amplifier Reference Voltage” is set to 2.55V.

Circuit Tab (right hand pane)

In WDS, click on the "Circuit" tab (right hand pane) and view the circuit diagram for the Type III compensator. Please study the circuit diagram and identify the location of the Type III compensator components.

Frequency Response (right hand pane)

Click on the "Frequency Response" tab (right hand pane) again to view the Bode plot. Then observe how the Bode plot changes as you type in the Nearest Preferred Values (NPVs) used in the actual circuit in to the relevant boxes in the "Controller Component Values" section of the "Analog (Non-Isolated)". The component values are shown below:



WDS will always use the user defined values instead of calculated values. So now that you have changed the component values to their NPVs, your crossover and phase margin will also have changed. The simulated crossover frequency and phase margin have changed because you have now entered the values of the components used on the actual PCB, which are slightly different to those calculated by WDS. Thus your compensator poles and zeros are slightly different from the ones calculated by WDS.

Power Loss Budget Tab (right hand pane)

This tab toughly estimates the total power loss in various components of the power supply.

Total power loss budget (PLB) is estimated based on the user's efficiency requirements and this is displayed at the bottom light hand side of the WDS window. A warning is displayed of the losses exceed the total PLB.

Summary Tab (right hand pane)

This is the final summary of your design. It includes all the parameters calculated by WDS including all the voltage and current stresses on the devices in addition to all the frequency repose Bode plots.

When your design is complete, you can save this as an rft file by pressing the "Save Summary As" button or insert on your own reports by simply copying and pasting.

Finally pressing "Generate and Email Transformer Design to" allows you to generate an email with all the parameters relevant to the transformer which you can send to your magnetics supplier of choice.

Spice Simulation Tab (right hand pane)

This is an extremely power full feature full explanation of the available simulations is given when you click on this tab.

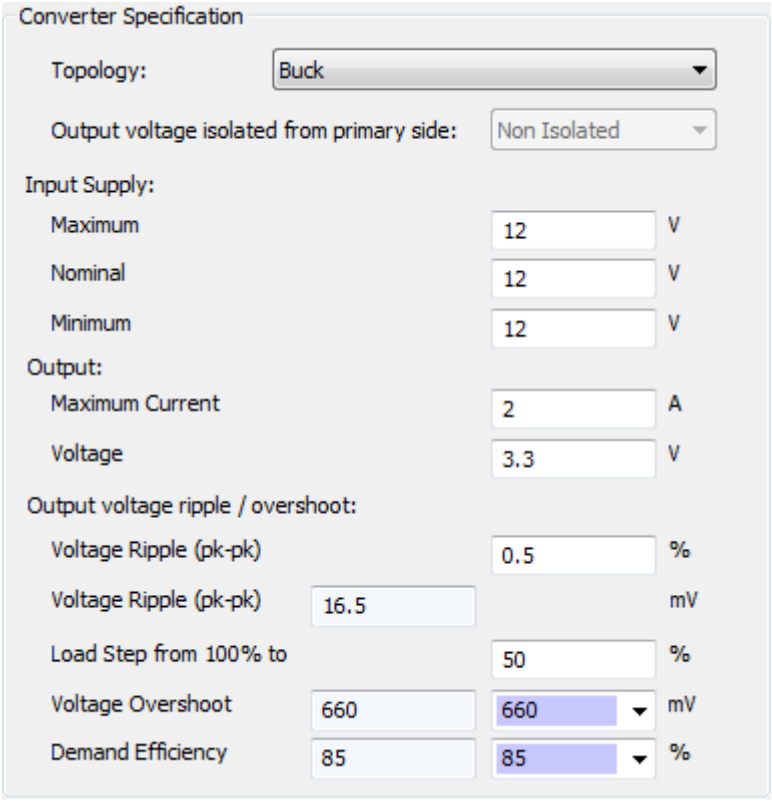
Design Example 2: Digital Voltage Mode Buck

Preamble

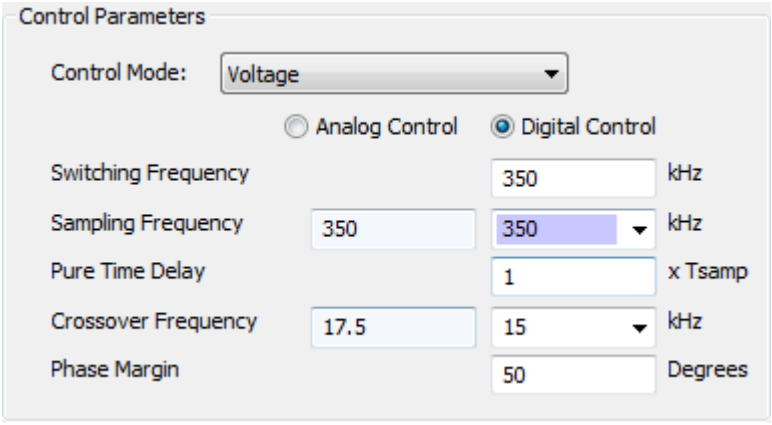
The design procedure is the same as Analog with the notable exception that instead of analog component values WDS will provide digital coefficients in various number formats.

Specification

On the "Specification" tab, enter the following specification into the relevant fields under "Converter Specification":



Again on the "Specification" tab, enter the following specification into the relevant fields under "Control Parameters":

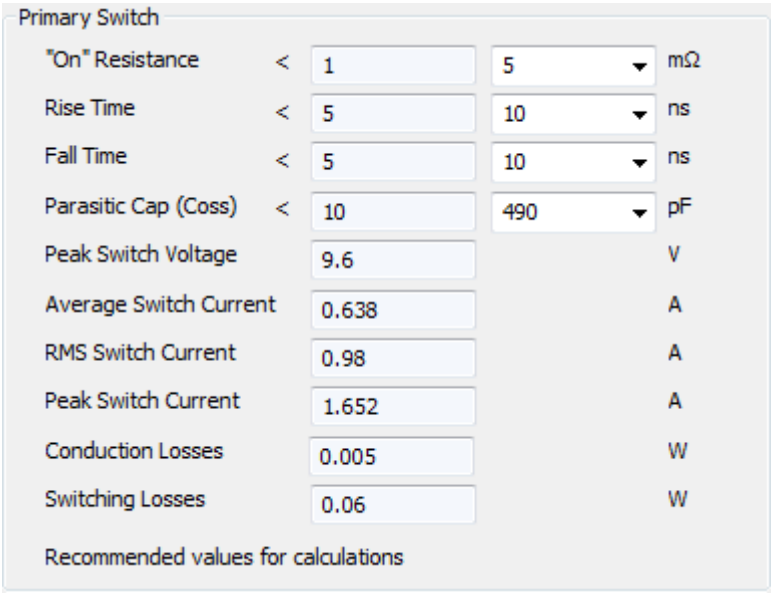


As you update the values, WDS will automatically update the design and show the associated Bode plot in the Frequency Response tab.

Please note that I your sampling frequency is slower than your switching frequency WDS will automatically take this in to account and you do not need to add this in the Pure Time Delay box.

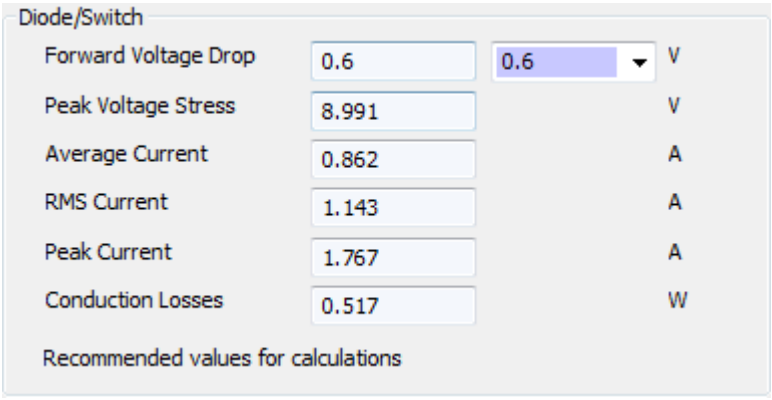
Semiconductors

Now move onto the "Semiconductors" tab, enter the following specification into the relevant fields under "Primary Switch"; these are the actual parameters of the FETs that we are using on the board. Important: for now, please ignore the values in the left hand column; these are calculated values based on your specification and will get updated as you input various parameters in the future tabs.



These are the values for the semiconductor switches that we are using on the Biricha Buck board used in our workshops.

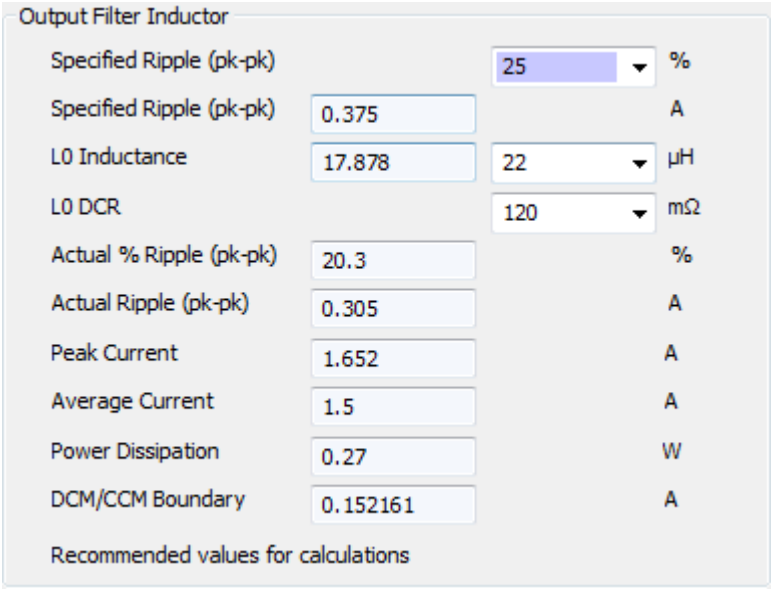
Again on the "Semiconductors" tab, enter the following specification into the relevant fields under "Diode/Switch":

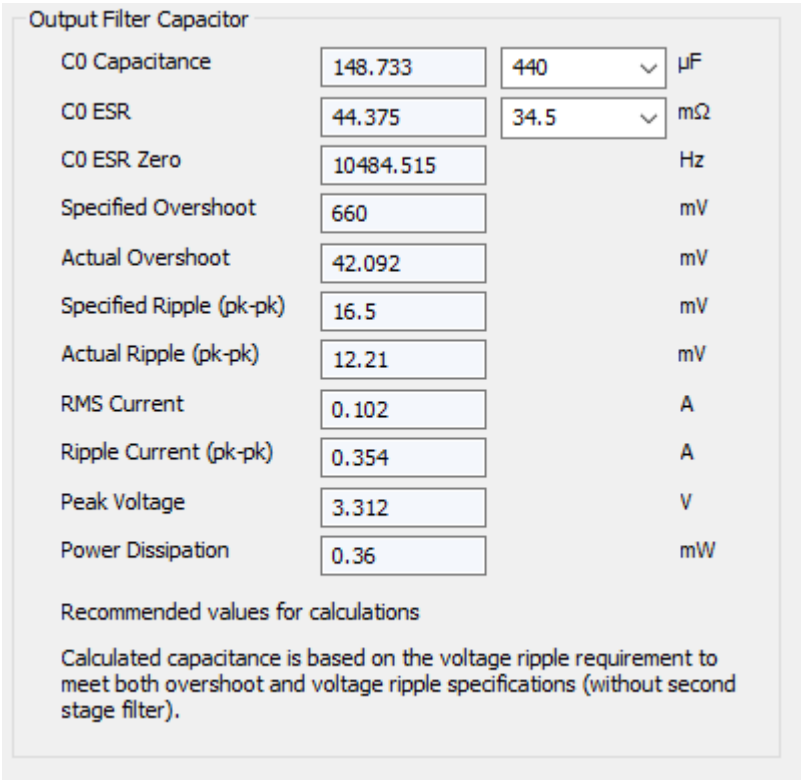


Choke/Output Cap

Now click on the "Output Filter" tab. On the "Output Filter" tab, enter the following specification into the relevant fields under "Output Filter Inductor ":

Again this is the parameters of the Wurth inductor that we are using on the Biricha Buck board. Also on the "Output Filter" tab, enter the following specification into the relevant fields under "Output Filter Capacitor ":

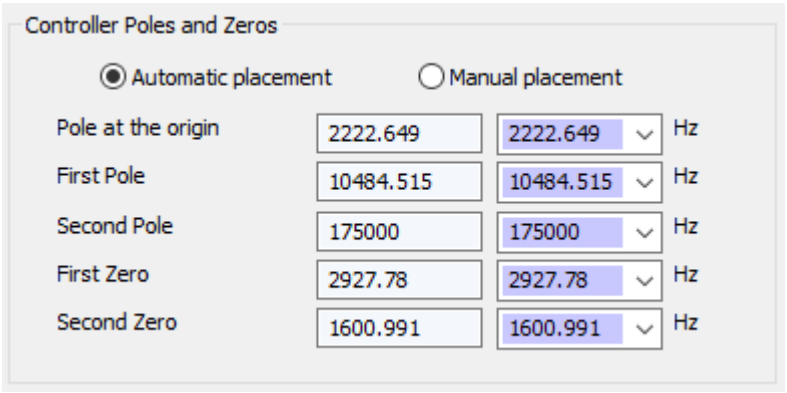




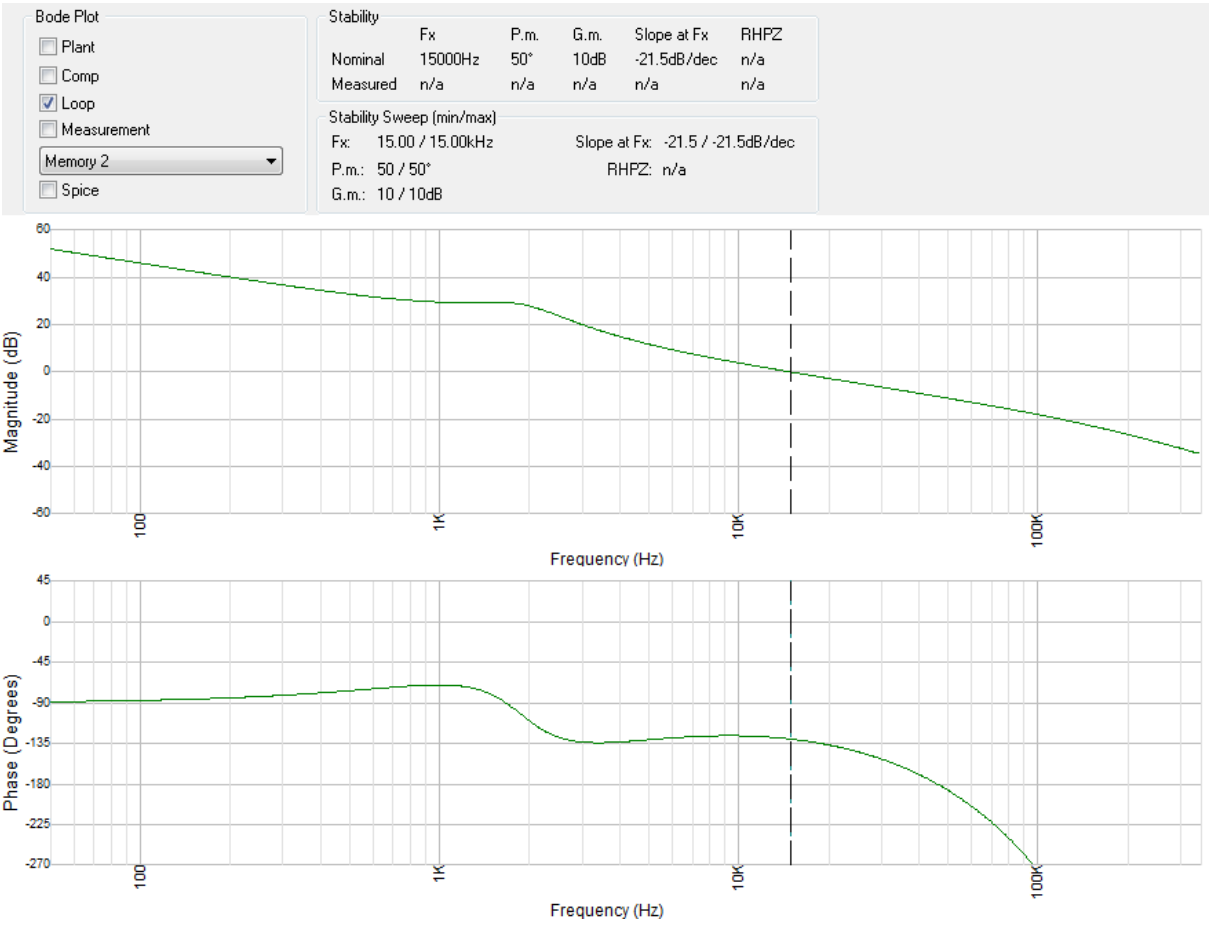
Note that we are using 2x220uF caps and we measured the ESR to be around 70mΩ for 1 capacitor. The ESR of 2 in parallel will be around 34.5mΩ.

Controller Design

On the "Controller Design" tab, WDS has selected a Type III compensator. The automatic pole/zero placement algorithms have calculated the location of the poles and zeros such that the loop will achieve the crossover frequency and phase margin specification concurrently, i.e. WDS has designed a compensator that will give a crossover of 15kHz and a phase margin of 50 degrees (after phase erosion). As shown below:

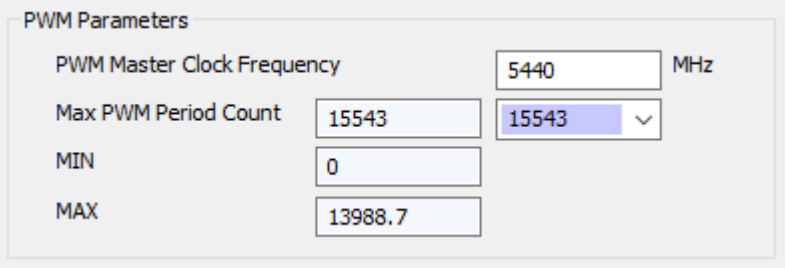


If not already in context, click on the "Frequency Response" tab on the right hand side to view the simulated Bode plot. Under the "Bode Plot" section of this tab, make sure that only "Loop" is ticked. This will display the simulated loop response. See that the simulated crossover frequency and phase margin are displayed at the top of this tab.

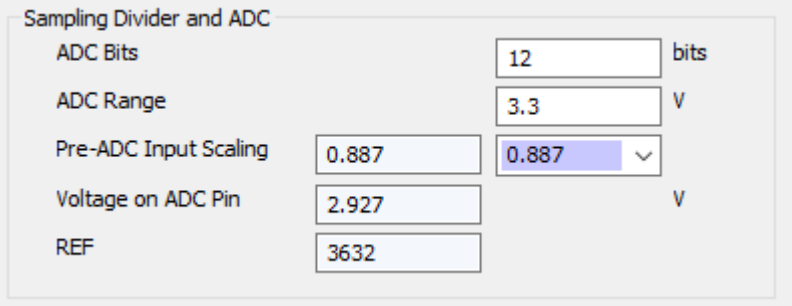


Digital (Non-Isolated)

Click on the "Digital (Non-Isolated)" tab, enter the following into the relevant fields under "PWM Parameters"



Again on the "Digital (Non-Isolated)" tab, enter the following into the relevant fields under "Sampling Divider and ADC":

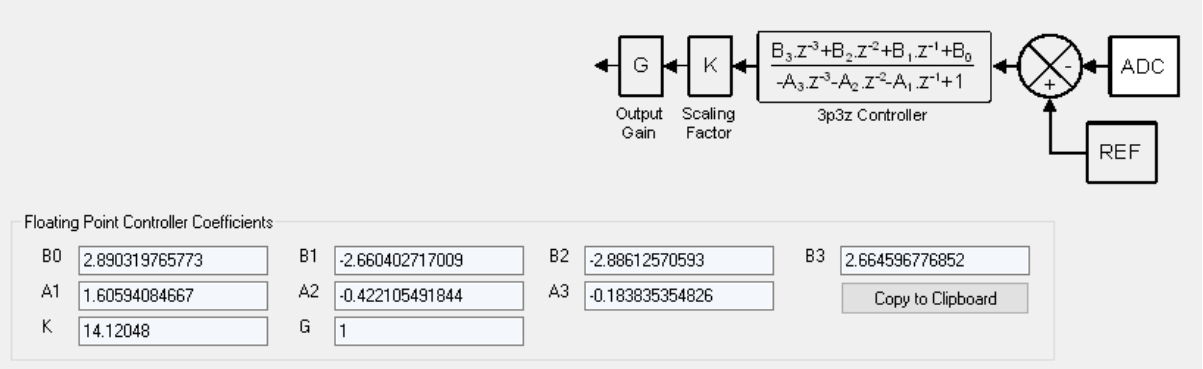


WDS will now be able to calculate the correct scaling factors for the digital controller.

Digital Controller Coefficients

WDS provides digital controller coefficients in floating point format suitable for the STM32 range of MCUs from STMicroelectronics. The general principle stays the same for all other coefficient formats.

Click on the "Coeffs (ST)" tab (on the right hand pane of WDS). WDS will then calculate the correct controller coefficients.



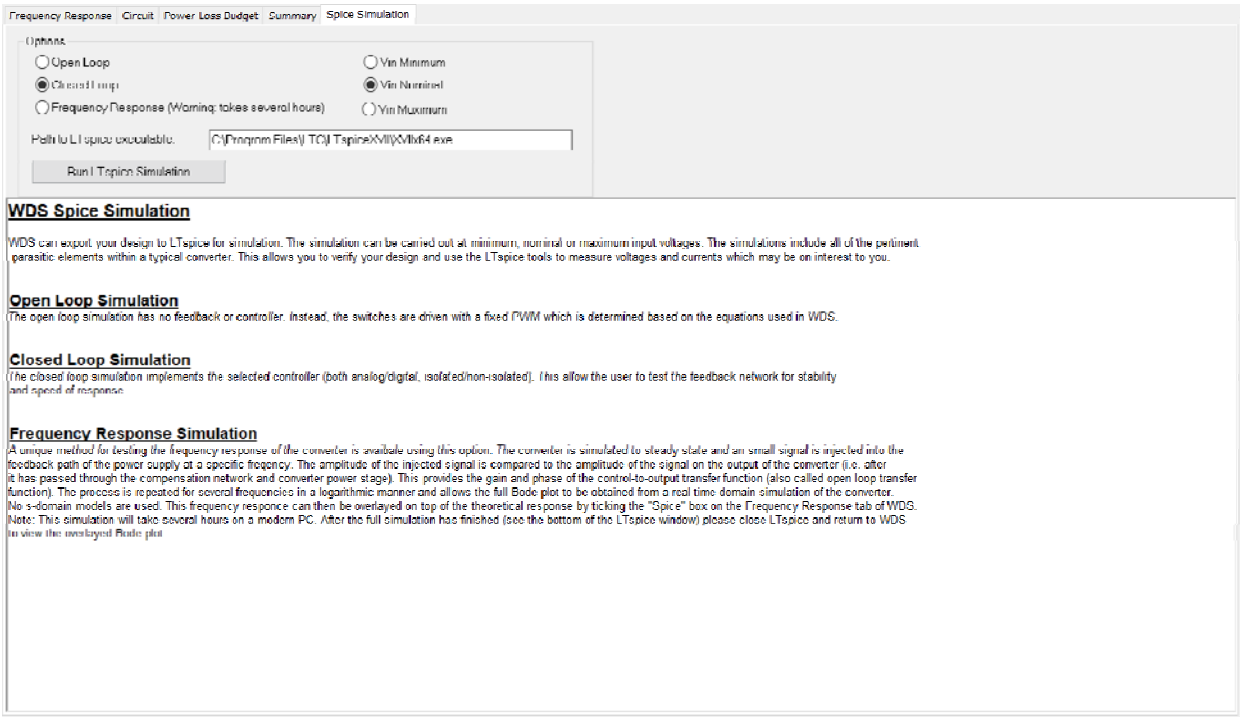
The user simply clicks "Copy to Clipboard" and then pastes the controller coefficients directly into their code.

Other Options

In this section we explain some other important options that may not have been covered with the above two examples

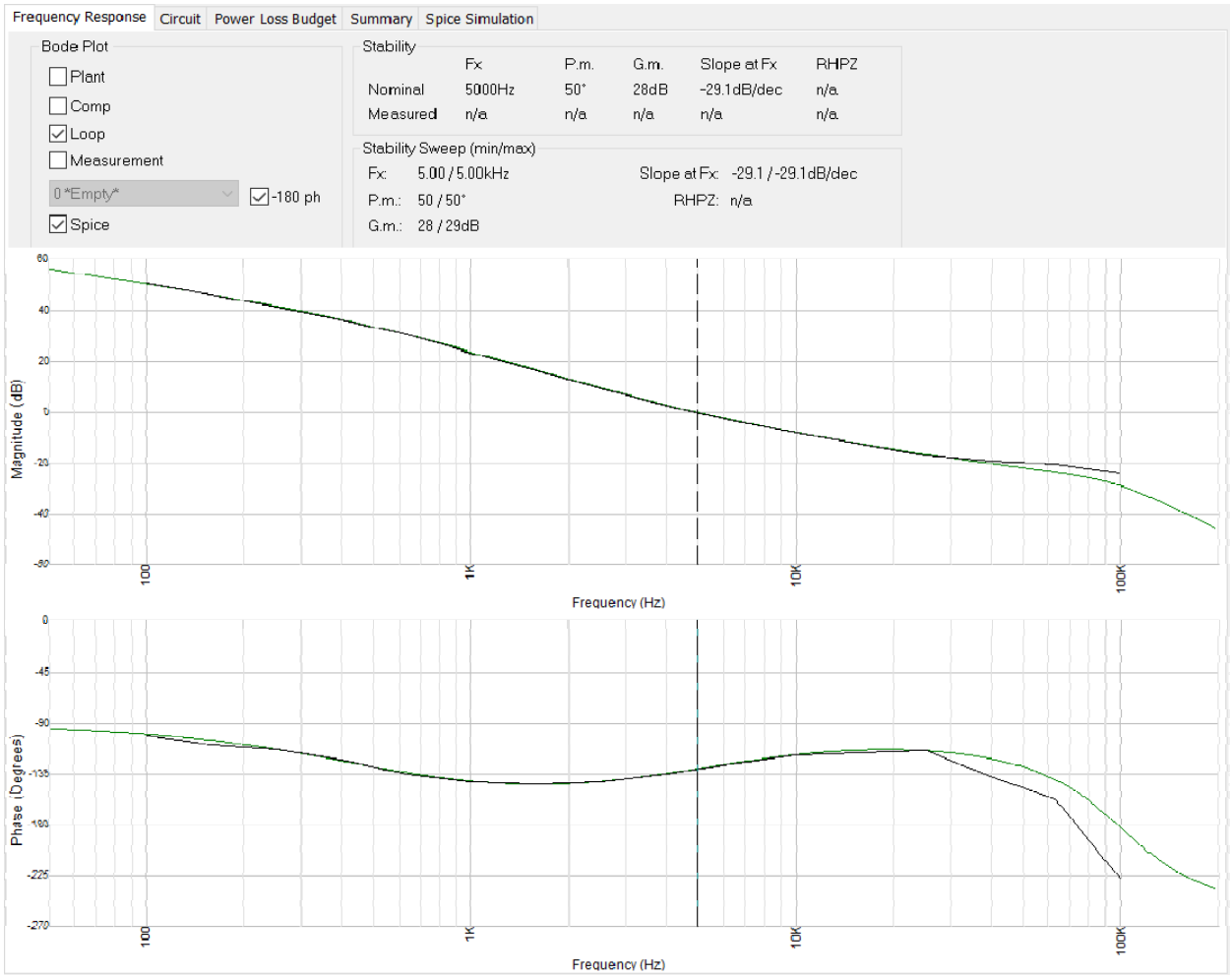
LTspice Simulations

WDS allows LTspice simulations in both time domain and frequency domain of your power supply, provided that LTspice is already installed. You can run a spice simulation by clicking in the “Spice Simulation” tab and then “Run LTspice Simulation” in the right hand window:



Important: to import the results of your LTspice Simulation back into WDS you must close LTspice first.

For example if you run a “Frequency Response Simulation” in LTspice by activating it through WDS, after the simulation completed you must close LTspice. You can then click on the “Spice” button on the Bode plot to display the LTspice. In figure below green trace is WDS output and the black trace is the LTspice output imported in WDS.



Current Mode Converters

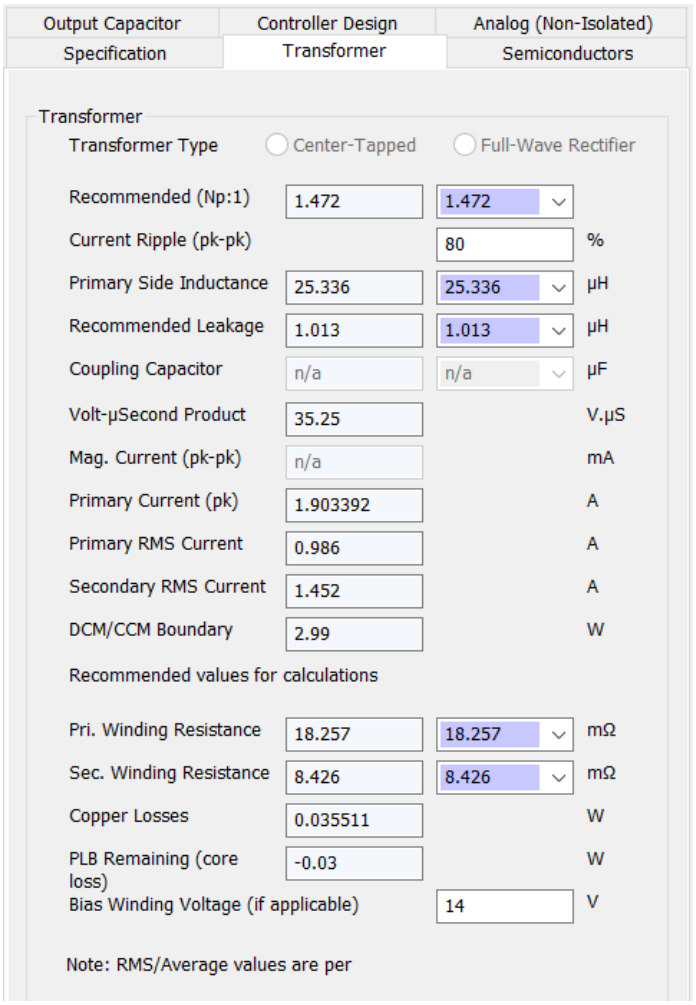
For Peak Current Mode, the options are almost exactly the same as voltage mode however in the “Controller Design” tab the user must enter the “Current Sense Gain” of the power supply. After that WDS will calculate how much compensating ramp is required and display in the “Amount of Ramp to Add” box. Please note that this is the height of ramp in volts if we were to allow the ramp to increase linearly for the entire Period T.

Isolated Power Supplies

You design an isolated power supply by simply selecting “Isolated” from the drop box in the Specification tab

Transformer

WDS provides most necessary information such as “turns ratio”, “Volts-us” product etc. so that you or your magnetics design can design you’re the magnetics WDS recommends the transformer turns ratio Primary inductance and leakage. The user can of course change these values after which all calculation will be based on the user’s input. The user can also indicate if a bias winding is needed by filling in the “Bias Winding Voltage” box. Once all design parameters are completed in this tab the user can email this specification to his/her magnetics provider by pressing the “Generate and Email Transformer Design to” button on the “Summary” tab.


Output Capacitor
Controller Design
Analog (Non-Isolated)
Specification
Transformer
Semiconductors
Transformer
Transformer Type
Center-Tapped
Full-Wave Rectifier
Recommended (Np:1)
1.472
1.472
Current Ripple (pk-pk)
80
%
Primary Side Inductance
25.336
25.336
µH
Recommended Leakage
1.013
1.013
µH
Coupling Capacitor
n/a
n/a
µF
Volt-µSecond Product
35.25
V.µS
Mag. Current (pk-pk)
n/a
mA
Primary Current (pk)
1.903392
A
Primary RMS Current
0.986
A
Secondary RMS Current
1.452
A
DCM/CCM Boundary
2.99
W
Recommended values for calculations
Pri. Winding Resistance
18.257
18.257
mΩ
Sec. Winding Resistance
8.426
8.426
mΩ
Copper Losses
0.035511
W
PLB Remaining (core loss)
-0.03
W
Bias Winding Voltage (if applicable)
14
V
Note: RMS/Average values are per

IMPORTANT NOTICE

All material presented in this document, associated software, exercises, appendices and any oral presentations made by or on behalf of Biricha Digital Power Ltd are for educational purposes only, are not intended for production or commercial applications and may contain irregularities and defects or errors. Furthermore, none of the material has been tested or certified by any government agency or industry regulatory organization or any other third-party organization.

Biricha Digital Power Ltd makes no warranties or representations, express, implied or statutory, including any implied warranties of merchantability, fitness for a particular purpose, lack of viruses, lack of patent infringements, accuracy or completeness of responses and lack of negligence. Prior to using, incorporating or distributing any of the material that has been included in this application note in any commercial product, you must thoroughly test the product and the functionality in that product and be solely responsible for any problems or failures.

In no event shall Biricha Digital Power Ltd, or any applicable licensor, be liable for any special, indirect, incidental, or punitive damages, however caused, arising in any way out of your use of the material presented in any Biricha Workshops, seminars or presentations. These excluded damages include, but are not limited to, cost of removal or reinstallation, procurement of substitute goods or services, outside computer time, labour costs, loss of data, loss of goodwill, loss of profits, loss of savings, loss of use or interruption of business or other economic loss.