White Paper

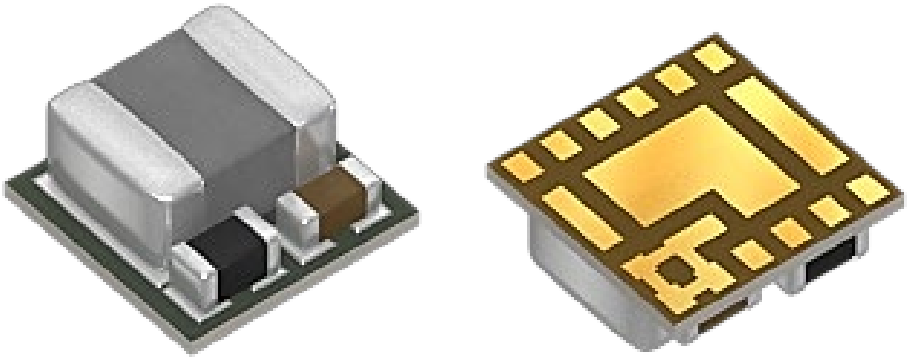
Designing with TDK's µPOL™ Modules

Introduction

TDK's new µPOL™ power modules (hereafter listed as µPOL) offer the industry's highest power density by utilizing advanced integrated circuit architecture and leveraging TDK's semiconductor embedded substrate (SESUB) technology. This combination enables 15W power modules to be achieved in 3.3 x 3.3 x 1.5 mm packages and 20W+ modules in a 5.8 x 4.9 x 1.6 mm package that may require additional input and output capacitors depending upon the end user's application.

To address thermal constraints typically associated with high power density, TDK's µPOL power modules make use of best-in-class patented thermal vias within the SESUB substrate. This structure allows for full rated current implementation without the need for external air flow.

This paper will introduce the reader to the readily available design tools and provide the steps needed to confidently design one's key power rails with TDK's µPOL power modules.



TDK-WP003
Aug/2022 Rev 0

Background

TDK's µPOL power modules come in various power capabilities and current ratings. Currently, the line-up consists of: FS1403 (5Vmax, 3Amax), FS1404 (3.3Vmax, 4Amax), FS1406 (2.5Vmax, 6Amax) and FS1412 (1.8Vmax, 12Amax). The FS1403/04/06 all are available in the same 3.3 x 3.3 x 1.5 mm package and are all rated for 15W, while the FS1412 comes in a 5.8 x 4.9 x 1.6 mm package and is rated for ~21.5W. As can be observed, the maximum output voltages are dependent on the maximum output currents.

The modules themselves are made up by the following subcomponents: Faraday Semi (a wholly owned subsidiary of TDK) power die, the embedding TDK SESUB substrate, on-board bootstrap and input bias ceramic capacitors and the power inductor (or 2 in the case of the FS1412). For ease of layout, the 15W rated modules all have common pin-outs and pad layouts.

The TDK µPOL modules are available with pre-programmed standard output voltages that reflect the more common voltage rails or in the case of the FS1406-0600 and FS1412-0600, as customer configurable output voltage devices. All of the standard module output voltages and respective part numbers are shown in Table 1.

V_{out}	TDK Part Number
3.30V	FS1403-3300-A
5.00V	FS1403-5000-A
2.50V	FS1404-2500-A
3.30V	FS1404-3300-A

V_{out}	TDK Part Number
0.6V to 1.8V	FS1406-0600-A
0.70V	FS1406-0700-A
0.75V	FS1406-0750-A
0.80V	FS1406-0800-A
0.90V	FS1406-0900-A
1.00V	FS1406-1000-A
1.05V	FS1406-1050-A
1.10V	FS1406-1100-A
1.20V	FS1406-1200-A
1.80V	FS1406-1800-A
2.50V	FS1406-2500-A

V_{out}	TDK Part Number
0.6V to 1.8V	FS1412-0600-A

Table 1 – Standard Vout values per module series and respective part numbers

From this table, one can see that the four-digit suffix, after the module series, defines the preset output voltage. For example, the FS1406-1000 has an output voltage of 1.0V (1000 mV).

Furthermore, by also looking at Table 1, there are two part numbers that provide for a range of the output voltages, 0.6V to 1.8V to be precise. These are the FS1406-0600 (1.8Vmax, 6Amax) and the FS1412-0600 (1.8Vmax, 12Amax) modules. These devices are configured so that a simple voltage divider resistor network is used to generate an accurate output voltage and is set by the

selection and placement of the top and bottom (FS1406-0600) resistors or just the bottom (FS1412-0600) resistor. This is, and will be, the more common design approach when using these specific power modules. By utilizing these non-preset devices, greater flexibility is afforded to the user. The recommended resistor values versus output voltages are provided in Table 2 for the FS1406-0600 and Table 3 for the FS1412-0600.

TDK recommends, for the FS1406-0600, that the designer places a 4.7 pF to 47 pF compensation capacitor in parallel to a R_{TOP} value of 40.2K ohms (Ω). The required resistor value for R_{BOTTOM} then becomes dependent on the desired output voltage and the R_{TOP} / R_{BOTTOM} ratio listed in Table 2. If R_{BOTTOM} becomes greater than 100K Ω, then TDK recommends using a lower R_{TOP} value of 4.02K Ω with a compensation capacitance value of 47 pF to 470 pF.

V_{OUT} (V)	R_{TOP} / R_{BOTTOM}	V_{OUT} (V)	R_{TOP} / R_{BOTTOM}
0.65	0.08725	1.20	1.04700
0.70	0.17450	1.25	1.13500
0.72	0.20940	1.30	1.22000
0.75	0.26175	1.35	1.30900
0.78	0.31410	1.40	1.39700
0.80	0.34900	1.45	1.48400
0.85	0.43630	1.50	1.57100
0.88	0.48900	1.55	1.65800
0.90	0.52400	1.60	1.74500
0.95	0.61100	1.65	1.83300
1.00	0.69800	1.70	1.92000
1.05	0.78500	1.75	2.00800
1.10	0.87300	1.80	2.09500
1.15	0.96000	1.85 to 2.50	please see note

Table 2 – FS1406-0600 Recommended Top/Bottom Resistor Ratio Versus Output Voltage

Note: Output voltages of 1.85V to 2.50V are possible but are dependent on the minimum FS1406-0600 “off time”. The off time is the shortest time for which the device can be switched off. Please contact TDK for support in setting output voltages in this range. For 2.5V output voltage, the preset FS1406-2500 part can also be used.

As shown in Table 3 for the FS1412-0600 module, the R_{TOP} resistor has already been set and only R_{BOTTOM} is needing to be selected and placed by the designer. This table reflects some of the more common output voltages. Since this approach is using the analog mode, output voltages of any value between 0.6V and 1.8V can be implemented.

V_{OUT} (V)	R_{BOTTOM} (KΩ)
0.72	21.00
0.85	9.76
0.90	8.06
0.95	6.81
1.00	5.90
1.05	5.23
1.10	4.75
1.20	3.92
1.50	2.55
1.80	1.91

Table 3 – FS1412-0600 Recommended Bottom Resistor Values Versus Output Voltage

It is always recommended that these suggested resistor divider values should be verified for one's final Vout by point-of-load measurements and adjust the resistor value if needed.

Due to all of the output voltage options, TDK has performed detailed and extensive design efforts, that provide all the needed design tools and enable the user speed-of-design assistance. These tools are: optimized standard baseline circuits with needed external passive components, schematic generation and design layouts minimizing board space, while still addressing thermal considerations.

Step 1: Choosing the best µPOL power module and approach

Typically, choosing the right µPOL power module would require doing a proper power mapping effort, thus identifying all voltage rails and their respective currents. This activity assists the designer in being able to easily identify which µPOL power module is the best choice for each targeted voltage rail. It also allows for common modules to be used in multiple locations on the same design to help reduce costs via economies of scale. Additionally, the use of the analog mode, voltage programming the devices is not required and is simply a change of a resistor to accurately set the desired output voltage.

For example, take the case of a FPGA VCC supply voltage at 0.85V and requiring 10A. From the previous brief overview of the TDK µPOL module series, the 10A requirement defaults to the FS1412 series that is capable of supporting 12Amax. Therefore, this application would require using a 9.76K ohm resistor as specified in Table 3 to generate an accurate and stable 0.85V output.

Step 2: Accessing TDK's design tools

TDK has created multiple design tools that will assist the designer in not only speed of design, but

also ease of design. To access the website location with the µPOL power module series, one can either use the website address at www.us.tdk.com/POL or the QR code shown in Figure 1.

Figure 1 – QR code for accessing TDK's website design tools

Either selection will take the user to a landing page that appears like the page shown in Figure 2.

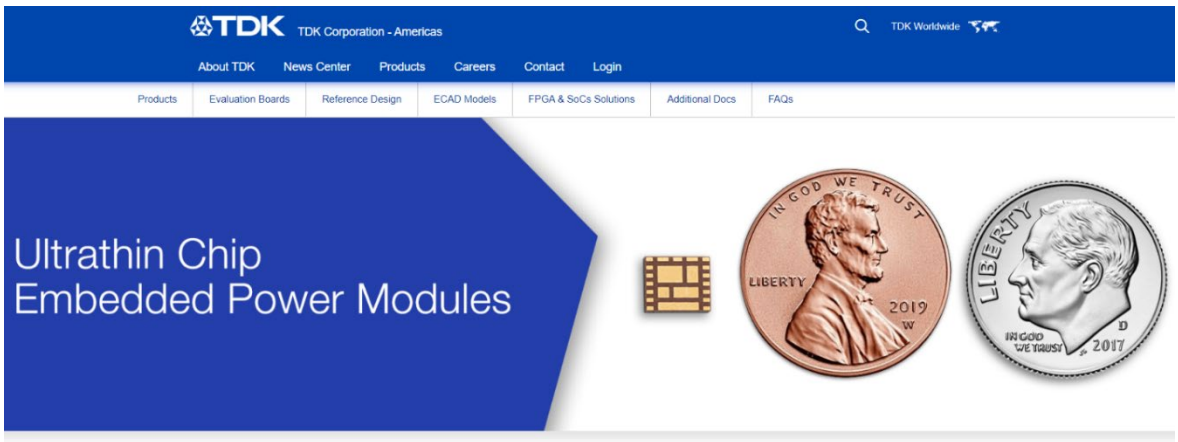


Figure 2 – TDK landing page for* µPOL *Starter Schematic design tools

At this point, one is ready to start accessing the design tools.

Step 3: µPOL Starter (Baseline) Schematics

If the designer is not already registered with Ultra Librarian, now would be a good time to do so. To access all FPGA/SoC and starter schematics, it is required that the user be registered and there is no charge to do so.

Once registered and back on the TDK µPOL landing page, there are two options for accessing the baseline reference designs. The first method is by scrolling one-third down through the landing page, until the "Evaluation Boards", "Reference Designs" and "ECAD Models" sections appear. Here the user can simply "click" on the "5W to 40W Multiple Modules" link. The second method is to click directly on the "Ultra Librarian" logo. These are both shown in Figure 3 and are encircled in red.

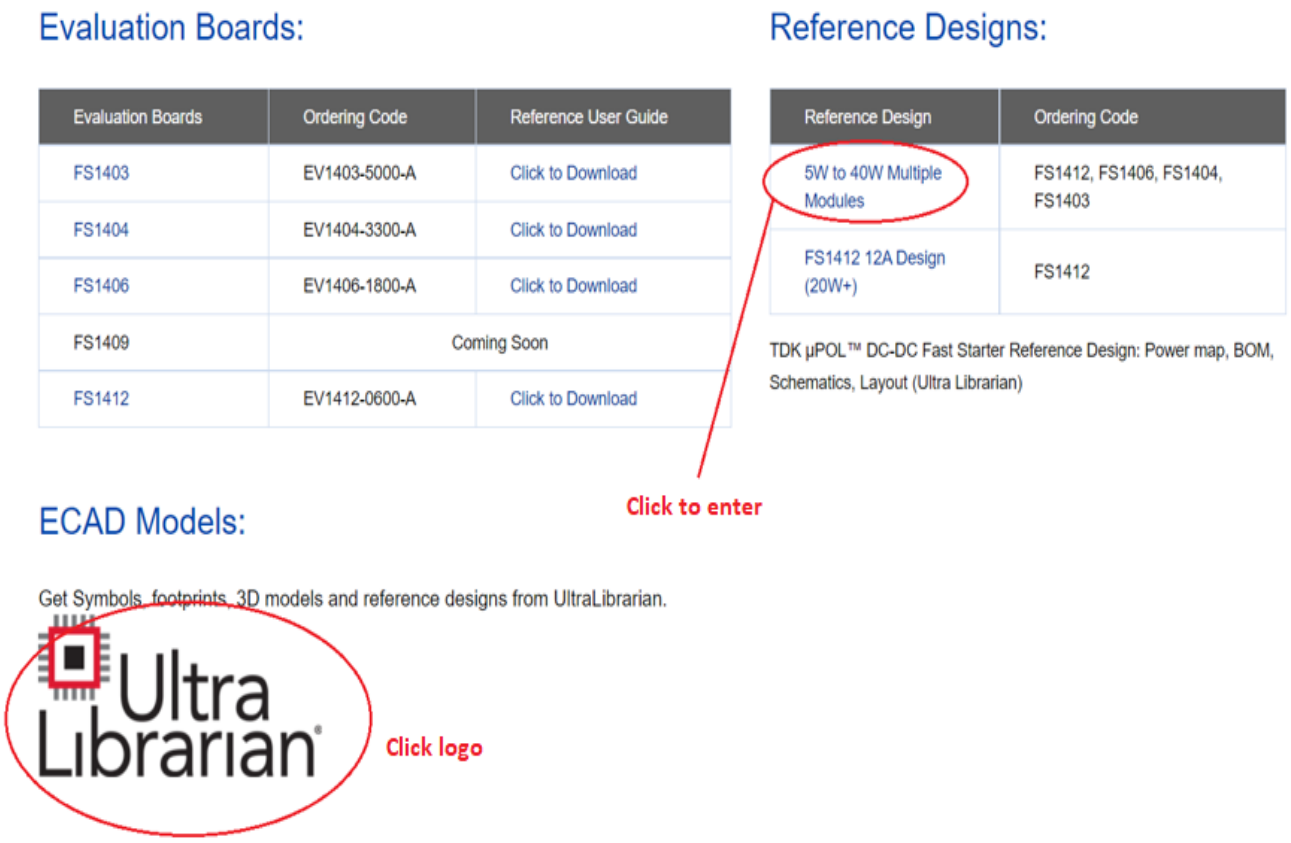


Figure 3 – Accessing the starter schematic reference designs information

By going through the "5W to 40W Multiple Modules" link, the user is directed to the following page shown in Figure 4.

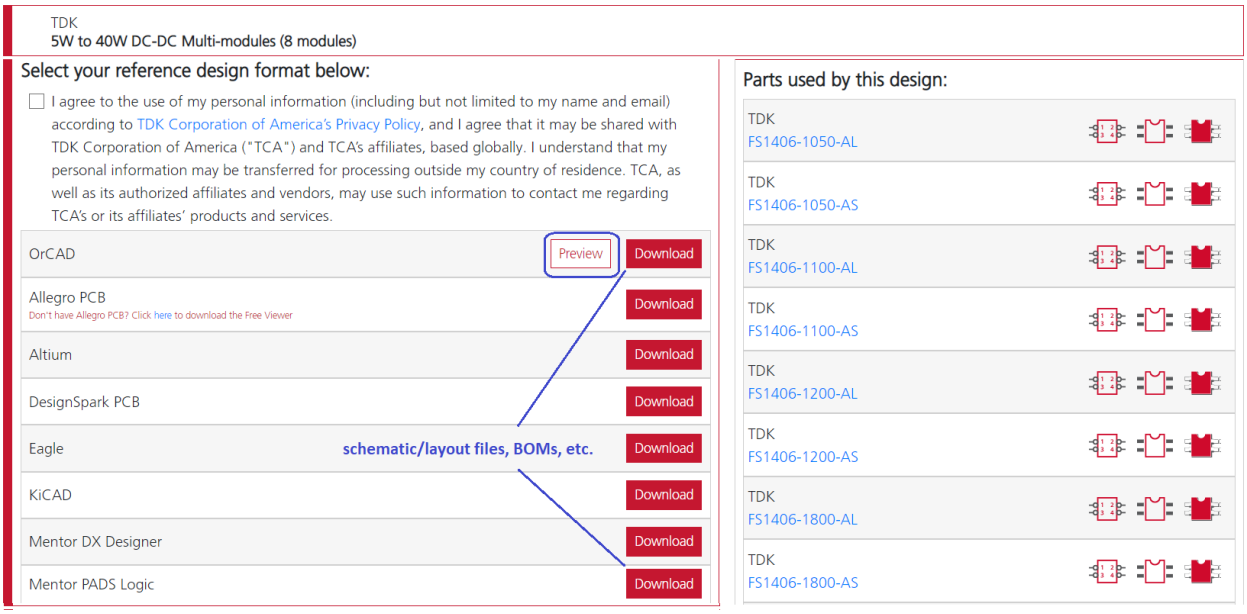


Figure 4 – 5W to 40W Multiple Modules menu

This menu allows the designer to preview the circuit schematics by clicking the "Preview" button in the OrCAD row. Also, on the left side of Figure 4, the designer can access software design tools from multiple CAD tool vendors and obtain [download] the schematic and layout files, Bills of Materials (BOMs) and other key files for multiple schematic design software vendors. Everything that is needed to quickly design, layout and specify all external components … is all there for easy access. On the right side, are links for the symbols, footprints and 3D models for each µPOL module used in the starter designs.

Figure 5 showcases the 8 different power module designs, for different voltage rails and output currents when viewing the "Preview" files. Listed are single designs using both the FS1403 and FS1404, four designs using the FS1406 and finally, two designs based upon the FS1412. It is of special importance to use the reference design shown in Figure 6. The key point of this design is that it uses the analog mode and has the output voltage divider resistors encircled in red. The reference designators for these are R80 (R_{TOP}) and R81 (R_{BOTTOM}).

For all the provided reference designs, the input and output capacitors values are listed. However, one needs to look at the BOM files for the capacitor technologies used, voltage ratings and temperature characteristics of the dielectric materials for each capacitor. Depending on the end application, some designs may require specialty capacitors with reduced ESR and/or ESL to provide energy during high di/dt transient events.

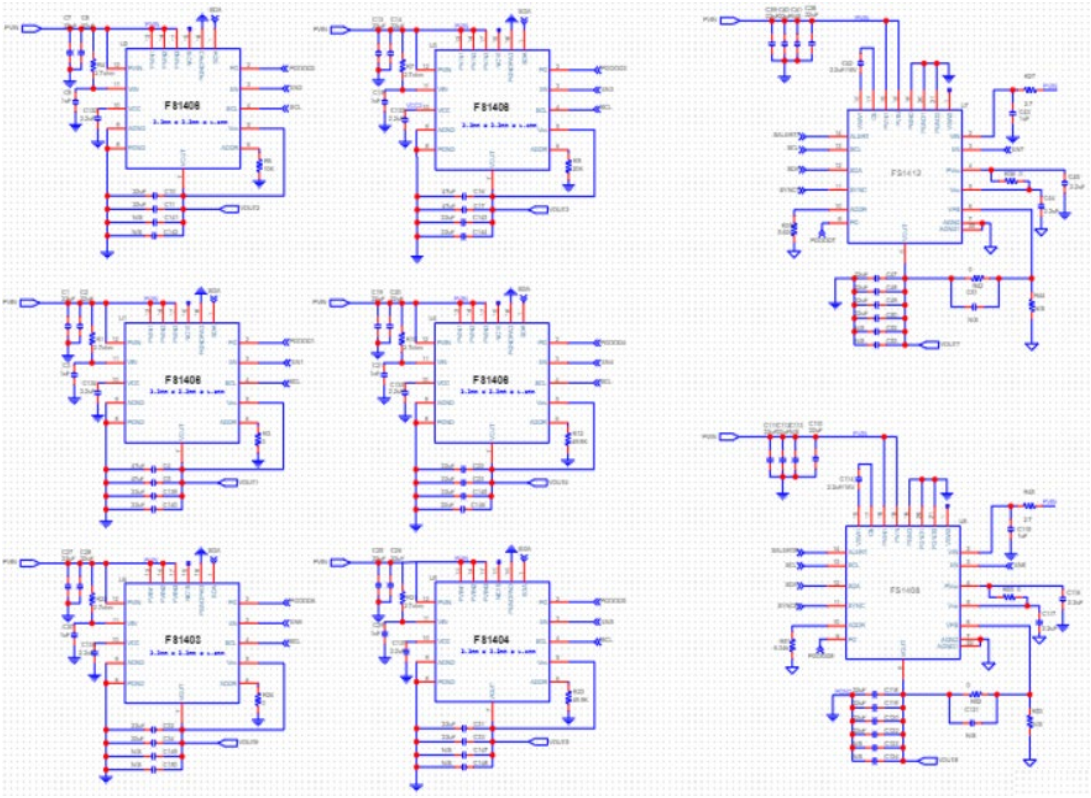


Figure 5 – 5W to 40W power strip evaluation board starter design schematics

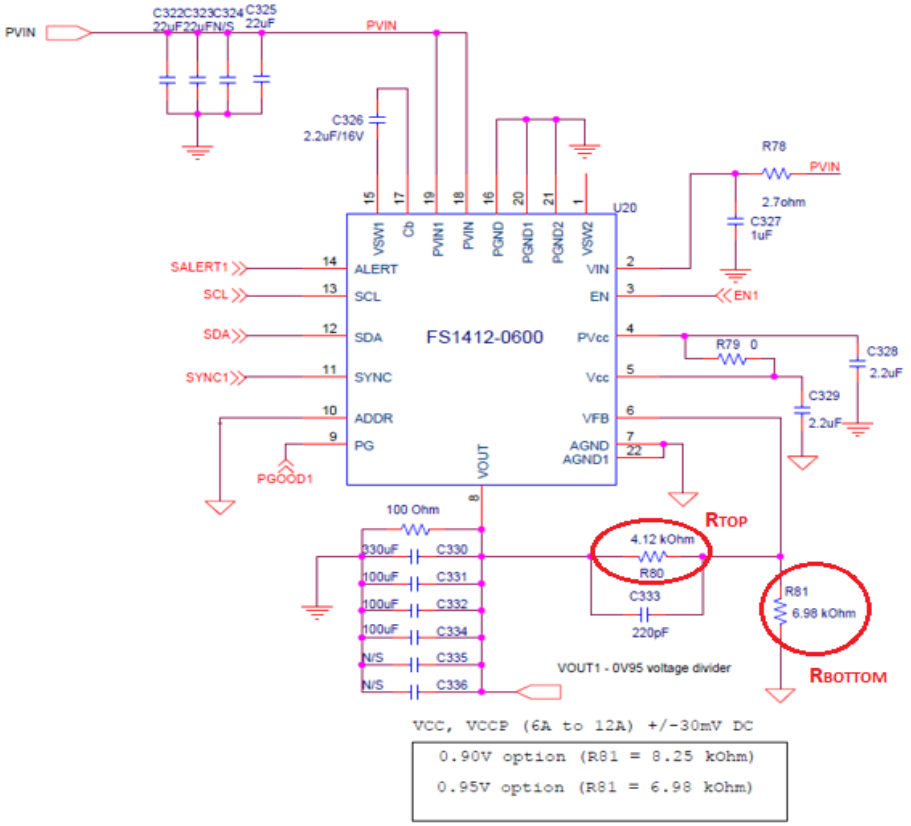


Figure 6 – FS1412-0600 12A design with output voltage divider resistors

Alternatively, by clicking on the Ultra Librarian symbol, the designer is directed to the "TDK Reference & Starter Designs" page as shown in Figure 7.

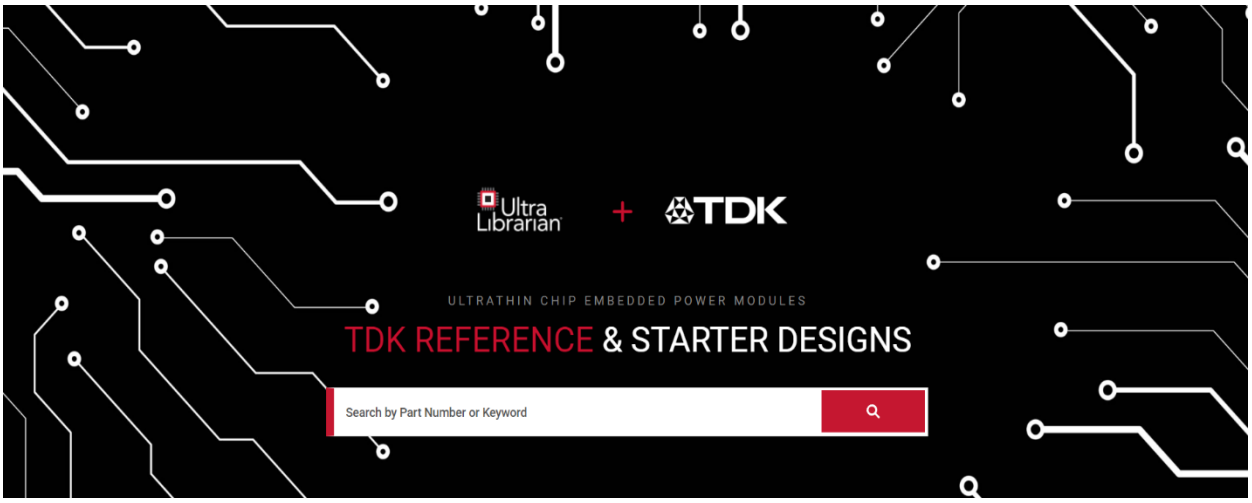


Figure 7 – Ultra Librarian web page

Once on this page, scrolling down to the next section allows for the same baseline reference designs to be accessed. This path, on the left side of the page, along with a 20W+ FS1412 reference design located on the right side, is shown in Figure 8.

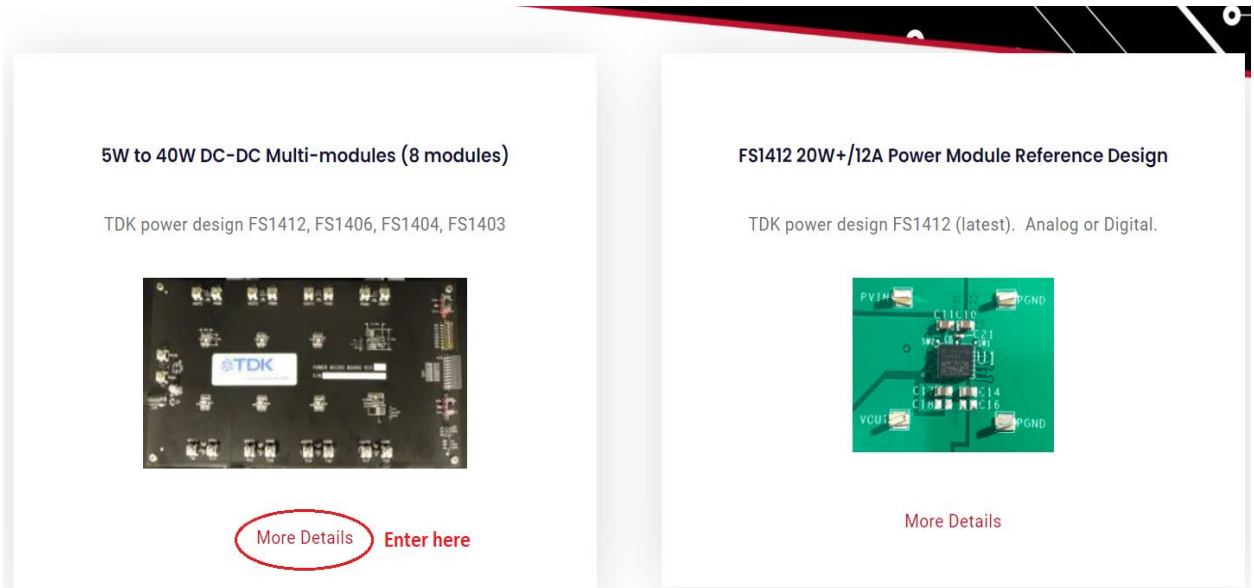


Figure 8 – Ultra Librarian path to reference starter designs

Regardless of the path taken to get to the starter designs, it is also important to know that reference designs are also available for many of the FPGA devices for a majority of the major FPGA suppliers. These designs address each of the key power rails, recommend the best module for that specific power rail and then provide the end solution design which has been thoroughly tested by TDK and the FPGA supplier.

Step 4: Adding external passive components

Once the starter schematic has been selected, the designer needs to determine the needed input and output capacitors. The starter schematic's design provides for adequate filtering and energy storage for most applications. However, it will be up to the designer to determine the actual capacitance needed (bulk capacitance for energy storage), the type of capacitor technology to be used and the allowable parasitic values of the capacitors in terms of equivalent series resistance (ESR) and equivalent series inductance (ESL). These parameters typically are dictated by the transient response requirements for di/dt.

Step 5: Verifying the final design and layout

Once a baseline design has been chosen and the corresponding layout has been implemented, there are still a few tasks that the designer needs to complete.

If the design requires support of a large di/dt transient event (change of current per change of time), then additional output capacitors may need to be added. The type of output capacitors may need to change based upon the power distribution network's requirements for the capacitor's equivalent series inductance (ESL) and equivalent series resistance (ESR).

A key aspect of the layout is the placement of the input capacitors. Placement of these close to the input voltage pin helps mitigate electromagnetic interference (EMI) which can create "noise" on the output voltage. For the output side capacitors, the distribution of these is important to minimize circuit board parasitics, which can impact the supply of stored energy (current) to quickly flow to the electrical load when needed.

Best design practices should also be followed when implementing connections to each of the power planes. Keeping these simple and clean ensure that there are no lengthy ground loops being created or schemes that also add a high level of parasitic inductance and/or capacitance.

Since the TDK µPOL power modules' current ratings do not require additional air flow, it is critical that the PCB thermal vias be designed, located, and enough in number to adequately support the system power while still maintaining thermal needs. Additionally, the copper weight on the PCB as well as the number of layers will also impact the thermal performance. Therefore, it is recommended that a thorough review be done on the proposed final thermal via solution prior to finalizing the end-product design.

Following the Ultra Librarian schematic guidelines and adhering to the suggested layouts, will allow for easy and quick design completion. Additionally, there is another key advantage achieved by following the suggested layout … board space savings.

A typical Ultra Librarian layout is shown in Figure 9. This example shows not only the simplicity of a design when using the µPOL power modules, but also the PCB real estate savings as well. Typical board space savings of between 25% to 50% is achieved by following the Ultra Librarian layout guidelines.

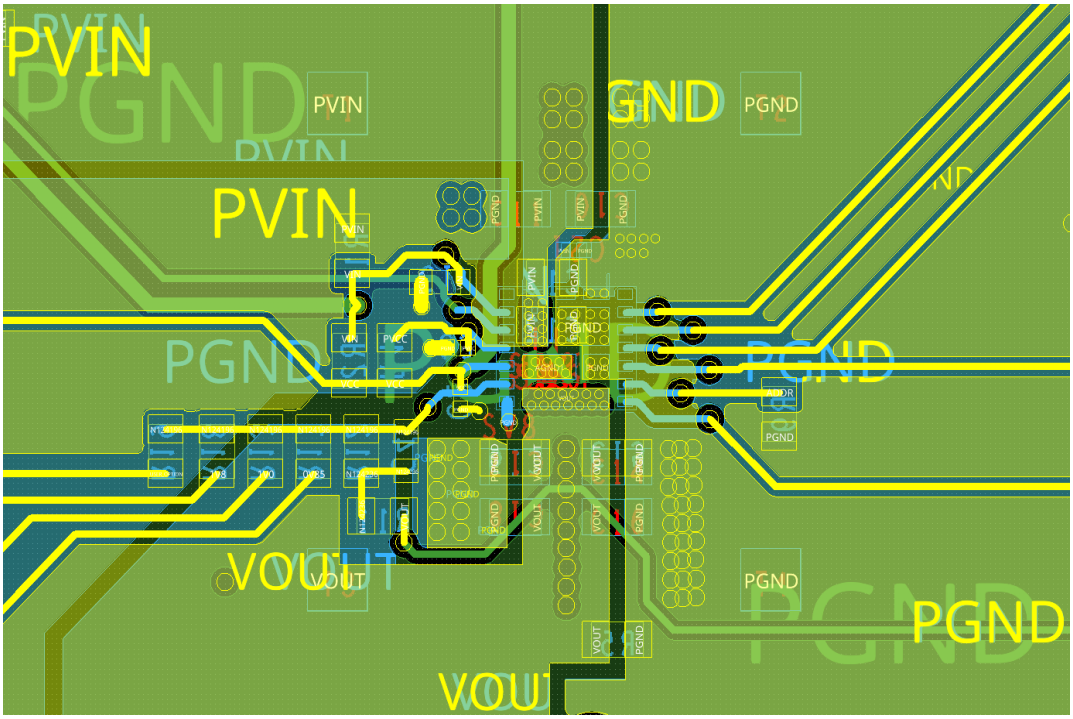


Figure 9 – Complete power rail solution layout example

And if the designer is not sure of their layout, TDK's µPOL FAE team can assist with a design review.

µPOL Design Steps Process Checklist

To recap what was previously discussed and to be used as a checklist, the following are the key steps when designing with the TDK µPOL power modules:

Define rails – It is always important to identify each power rail in terms of its voltage and maximum current requirement. Consolidating these into a power map [matrix] helps the designer to easily identify common parts and where the same µPOL power modules could be used and have the output voltage easily set by voltage divider resistors.

Select modules – The required output voltage may be the driving factor in determining which µPOL power module must be used. The FS1403 has an output voltage range of 3.3V to 5.0V, the FS1404 a range of 2.5V to 3.3V, the FS1406 is 0.6V to 2.5V and finally, the FS1412 with its 0.6V to 1.8V capability. There are some voltage overlap values between modules and the module choice will then be determined by the maximum current required for each power rail.

Register with Ultra librarian – This is a free service and must be done prior to accessing the starter designs. One can register at https://www.ultralibrarian.com/.

Access and leverage starter designs – The existing proven starter designs can be accessed either by going to www.us.tdk.com/POL or using the QR code shown in Figure 1. Once on this page, access the baseline designs by clicking on "Reference Designs" or the Ultra Librarian logo. From here, the user will navigate to the CAD options, login and download the baseline designs.

Access and leverage layout – The layout for each design is also accessed and obtained on the same page and in the same files as the schematics.

Check latest datasheet for schematic updates – Designs are constantly being improved, modified for more flexibility or optimized, etc., therefore TDK recommends obtaining the latest datasheets for up-to-date changes.

Add needed input and output capacitors – The needed input and output capacitor values and type need to be determined by the designer. The capacitors provided in the baseline solutions address typical requirements and, though they have margin built in, still need to be thoroughly evaluated if Vin is noisy or unstable and/or if Vout supplies a high transient load.

Chose and install output voltage divider resistors – As provided in Table 2 and Table 3, TDK provides pre-determined voltage divider resistor values to further assist in the ease of design.

Ground I2C and PMBus pins – To ensure proper performance, it is recommended that the input communication pins be electrically grounded on the PCB design if I2C or PMBUS are not used in the application.

Ensure thermals are addressed – verify PCB copper thickness is adequate, consider the number of PCB layers, and finally, the location, number and size of vias used within the thermal path.

Contact TDK for schematic and layout review – TDK's technical staff will review each design's schematic, layout, capacitor solution and resistor values as a complimentary service.

Summary

This paper has provided the reader an overview on where to find the design tools to easily and quickly design power rails based on TDK's µPOL power modules. These tools provide ready-to-go designs and generate the corresponding schematics, board layouts, the Bills of Materials and other needed files. All that is required by the designer is to finalize the input capacitor solution,

ensure that the output capacitor solution is adequate to handle worse case transient events and to install the appropriate voltage divider network resistor(s) when setting the output voltage via “analog mode” for the FS1406-0600 and FS1412-0600 devices.

Note: Some designers who are more familiar with the TDK power modules may want to explore and utilize the “digital mode” to accurately set the output voltage. The FS1403/04/06 can have their output voltage levels programmed via I2C communications while the FS1412 can be programmed using the PMBus. TDK’s technical staff can provide assistance if the “digital mode” is the voltage setting approach to be used. However, TDK recommends using the FS1406-0600 and/or FS1412-0600 as baseline products and using the “analog mode” to set the output voltage as needed.

TDK Product Center Website

Learn more about the broad portfolio of TDK products

Release 2022-08