How to Convert a UPD Package Substrate for Use with APD and SiP Layout

Product Version SPB16.6
March 6, 2013

Contents

Purpose

This document describes the procedure to import UPD spd2 format files in Cadence APD or SiP Layout tools and creating your APD/SiP Layout libraries based on imported data.

Audience

This document is intended for users who would like to migrate from UPD package layout tool to Cadence APD or SiP Layout Design tools.

.

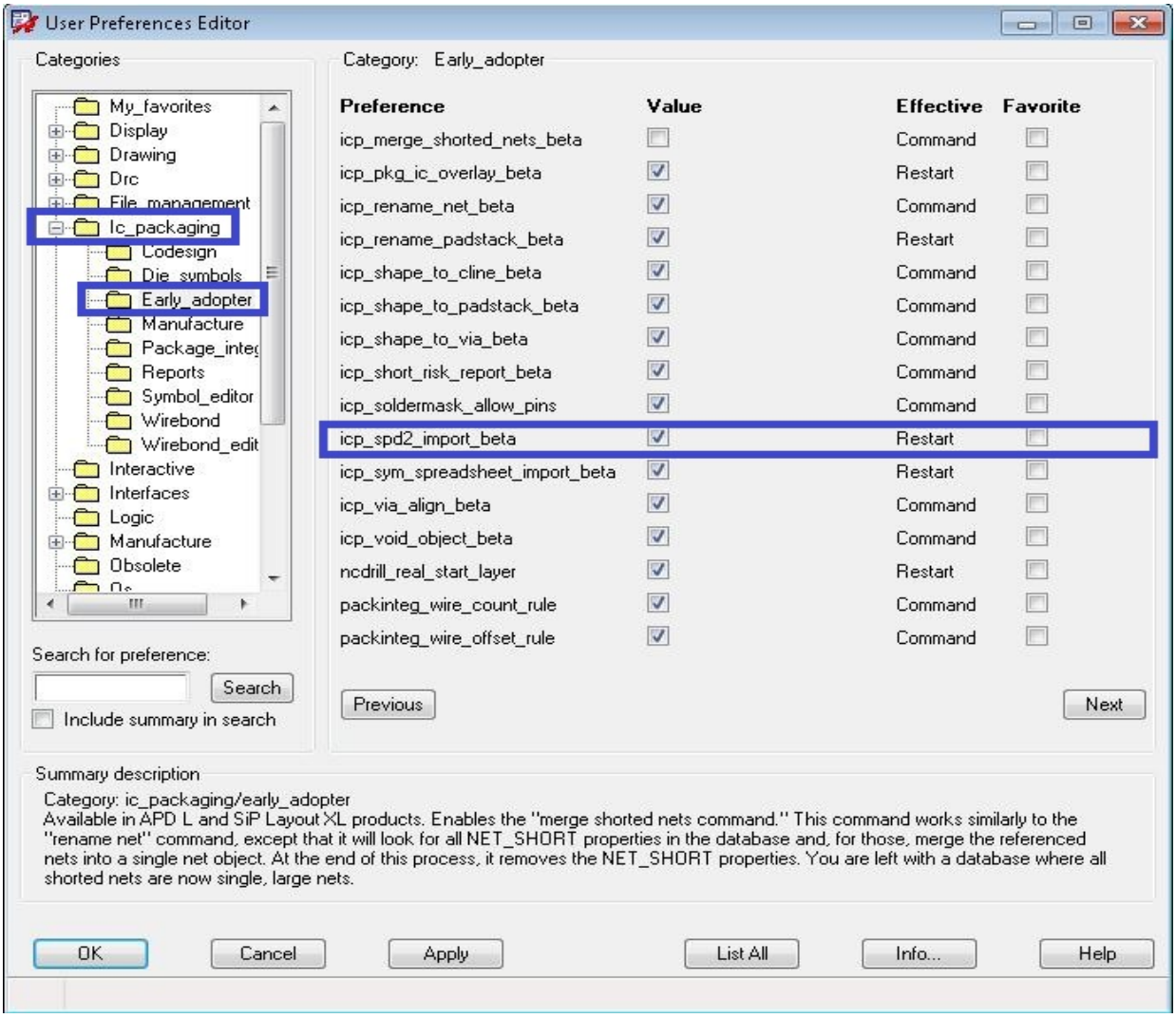
How to Convert a UPD Package Substrate for Use with APD and SiP Layout

If you are a user migrating from the UPD package layout tool to the Cadence APD or SiP Layout design tools, you will need to convert your existing designs into the new database format. You may also need to create libraries of symbols, padstacks, device files, and technology files.

One important thing to understand before continuing is that only NA2 and SPD2 formats from UPD – the ASCII text versions of a UPD design database – are supported for import. If your UPD design is currently saved in binary UPD (*.upd) format, you will need to open the design in UPD and save it to the SPD2 format. Also understand that SPD2 and SPD file formats are completely different. Importing an SPD file is not supported.
In this document, we will cover the import of a UPD “SPD2” database into APD or SiP Layout. The process in either tool is identical. So, any command menu locations, forms, log files, and other items mentioned should be the same in your APD tool as in the SiP Layout screenshots shown. The primary difference being any reference to die stacks, which are present only in the more advanced SiP Layout tool.

Procedure 1: Importing a Package Substrate Design

Before beginning, check the format of the database file you are importing. NA2 files may be imported with the default configuration of APD or SiP Layout. If you have a more recent SPD2 format file, you will want to enable this format's support in Setup -> User Preferences… ("enved") as highlighted below. After setting this variable, restart the tool.

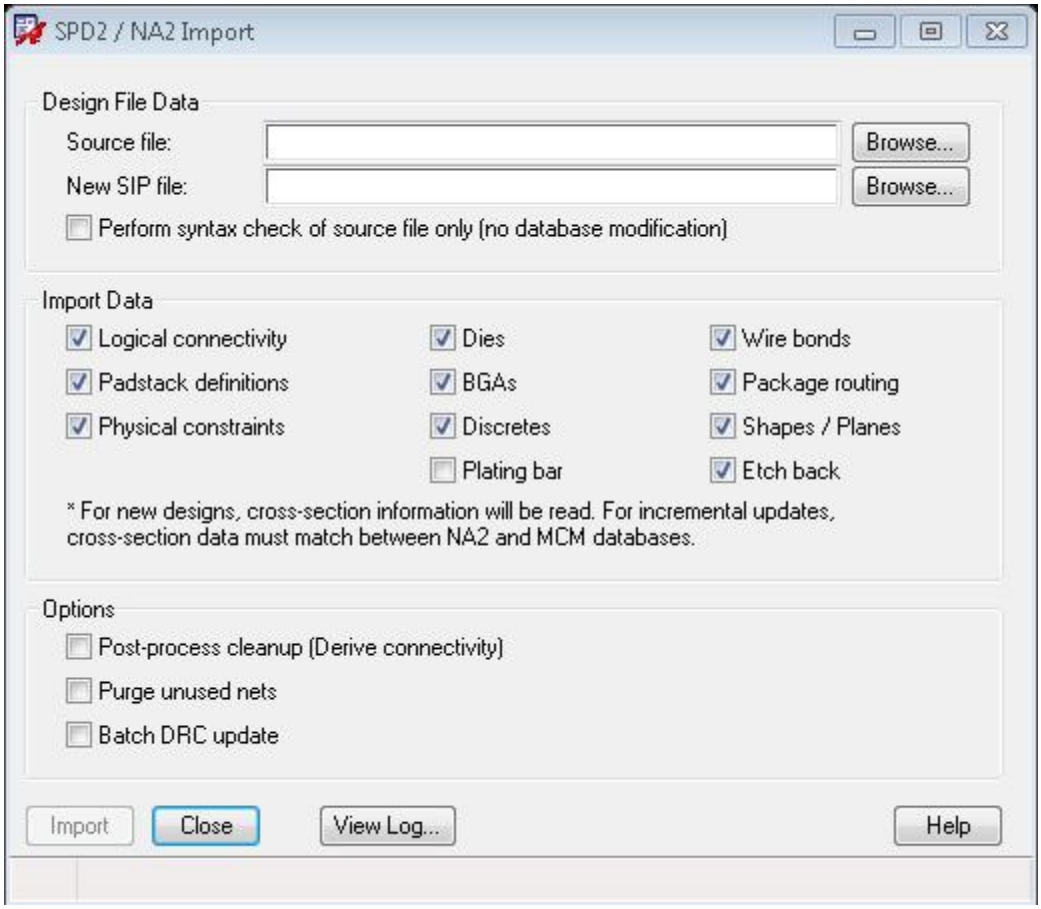


Secondly, if you are new to APD and SiP Layout and have not configured your library of materials, you should do so now. This information is not stored in the design databases themselves, but instead resides in the "mcmmat.dat" file, which you can find in the share/pcb/text subdirectory of your Cadence software installation.

Instructions and file format information for the materials file are located at the top of the file itself, which can be opened in any text editor. Since this one file is used to store all material information for all your designs, you should update it with all materials that you use in your designs. This assumes that you have a listing of these from your

manufacturing partners. If you do not, when you import a UPD database, you will get warning messages for materials that are not found in this file and that you will need to add. In order to avoid overwriting materials information referenced by other designs you may have, the library materials information is never directly updated based on the UPD database being converted.

You are now ready to begin. Run the File -> Import -> SPD2 / NA2… ("na2 import") command. This will bring up the import configuration form, below, where you can choose what specific data you want to import. The image shown illustrates the default settings that Cadence recommends for your database conversion.



After browsing to your source UPD database file and specifying the default file name for the converted database (this will always default to the same file name and location as the UPD database, but with the new file extension), set the data to import.

Note: *Because of the large differences in how plating bars are defined in terms of database structures between UPD and APD / SiP Layout, Cadence does not advise importing the plating bar through this process. Commands are available inside of the tools to generate the plating bar after import is complete and verified.*

Procedure 2: Identifying Data Requiring Manual Conversion or Import

Once you have finished importing your design file, it is time to make any necessary updates for information that is not 1:1 compatible between the two environments, that may not be available in the SPD2/NA2 file itself, or that may be exclusive to the APD and SiP Layout environments.

The first step in doing this is to check the import log. This provides detailed information on any data that could not be read, data which may be missing, and any advisories that you should be aware of. It also includes a complete listing of the constraints which have been mapped from the UPD database to the APD / SiP Layout database.

Note: *If you feel a constraint mapping is not present that should be, please contact your customer support representative with what you feel that mapping should be.*

Should you see any errors or warnings in the log file, they will almost always include a reference line number inside the source file. From that line number, you can get highly specific information about the affected object, as the SPD2 and NA2 files are both ASCII text files which you can open in any text file editor of your choice. It is recommended that you deal with these warnings and error messages as a very first step. They may indicate common issues that will arise in future design conversions and therefore will warrant a note for future reference.

After processing any logged issues, it is time to move on to information specific to your designs. The following is a list of common items to check. It is not exhaustive as your designs may differ in structure and definition of other users. This is a guide only, and it is advisable that you create your own checklist based on your specific flow and data.
Common items requiring manual data entry, object creation, and setting verification:

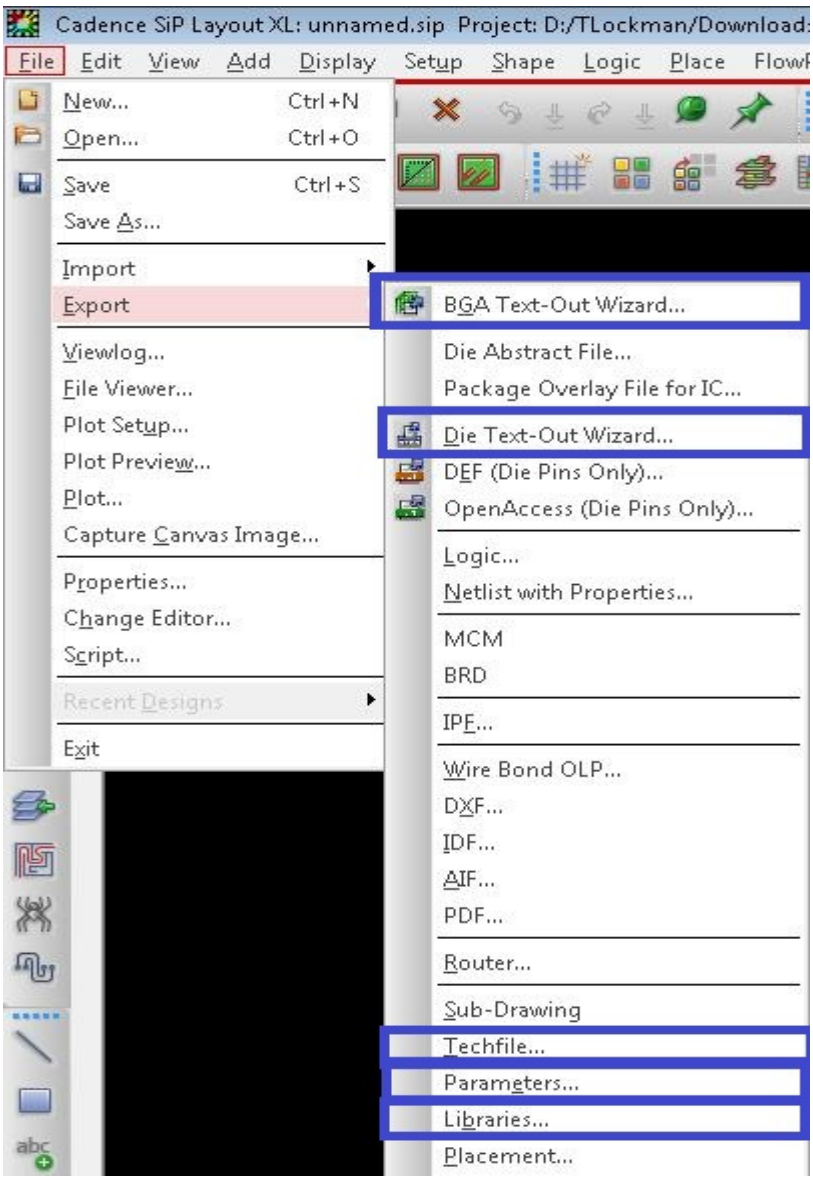
- **Plating Bar** – As stated in procedure 1, it is not recommended that you import the plating bar from the UPD design, as the two design environments handle the plating bar in very different manners. Instead, after conversion, run the Manufacture -> Create Plating Bar… (“pbar create”) command. This will generate the plating bar automatically based on clines extending beyond the package edge.
- **DRC Constraints** – Only constraints with a clear 1:1 mapping between the two environments are translated. Some items exist in UPD which are not present in the APD/SiP tools and vice versa. Others may exist in both environments but be applied differently (e.g. an object type constraint versus a global database constraint). Use the Cadence Constraint Manager to configure any constraints which have not yet been set up.
- **Net Shorts –** If you design contains nets that are shorted together (typically done using UPD’s “super nets” capability), these nets must be manually shorted together inside of APD / SiP Layout. Use the Route -> Define Short… (“define shorting scheme”) and Route -> Create Short… (“create short”) to configure the legal net shorting schemes for your design.

- **Static vs. Dynamic Shapes –** In order to preserve their exact boundaries, all shapes imported from a UPD database are initially created as static shapes. This means that any changes to their boundaries, including voids around objects on other nets, must be done manually. Consider converting shapes to dynamic mode to allow real-time voiding as you continue to work on the design.
- **Artwork Films –** If you use Artwork films for generating manufacturing outputs, you will need to define your films using the Manufacture -> Artwork… ("artwork") command. These film definitions are used by select other commands, including PDF file generation and film area reports. If you will be using any of these tools, you will also need to configure films at this point.
- **(SiP Layout Only) Cavity, Die Stack, and Die Physical Properties –** If you are using SiP Layout, use the Edit -> Die Stack… ("diestack editor") to verify cavity settings as well as die ordering within the stack. Finally, for any flip-chip die components, configure the solder bump geometries for the pins.

Once you are happy that your design represents a 100% accurate conversion, consider whether any of the objects should be turned into library objects for future design reuse. If this applies to you, carry on to procedure 3. If not, skip ahead.

Procedure 3: Building up your APD / SiP Layout Libraries

Now that you have finished importing your design, you may wish to export specific information to build up your library of reusable design elements. This will give you a quick jump start when beginning similar new designs. There are many different aspects of your design which can be converted into library elements. The specific commands that you use to do this, however, are all found in the same menu of the tool: The File -> Export menu, as shown below:



Physical Library Data

Physical information, whether it be a library symbol, padstack definition (and corresponding custom pad shapes), mechanical symbols, or even component device files, are all exported with a common command. The File -> Export -> Libraries… (“dlib”) command manages this for you.
The only exception to this is a choice that you have. Do you want your library objects for BGA and Die components stored in binary database format (*.dra) or in a readable spreadsheet format that you can look at and modify in any spreadsheet or text file editor? If you prefer the text spreadsheet format, use the File -> Export BGA / Die Text-Out Wizard… commands (“bga text out” and “die text out”, respectively) to write these files out. DRA files may contain more customized information such as logos and non-rectangular component outlines, while TXT files are easier to read and need no uprev/downrev as you change Cadence software releases.

Technology Data

Design rules are stored in technology files. To export the tech file for your design, use the File -> Export -> Techfile… (“techfile out”) command. This simple command requires only that you provide a file name and location for the exported techfile. Later on, you can read this techfile’s information into a different design using the File -> Import -> Techfile… (“techfile in”) command. If you want a quick report of the differences between a library techfile and your current design, use the Tools -> Technology File Compare… (“techfile compare”) tool.

User Preference Data

The final types of data you can export for library or personal reuse are the various design parameter settings. From color palette values and preferred layer/wire profile/component color assignments to text sizes and default behaviors and settings for certain commands, these settings are managed with the File -> Export -> Parameters… (“param out”) command.

With this command, you need only specify the file name along with what parameters you want the file to contain. Import this into another design file with the File -> Import -> Parameters… (“param in”) command to quickly make any design conform to your ideal look and behavioral feel.

Did You Know...

Performing these tasks all at once, by one person (or by following the flow documented by a single expert in your company) can make the process efficient and minimize errors. This one expert will understand any additional data which may need to be manually imported after translating the core design. As every company’s designs have their own unique characteristics, the specific things to look for and manually update will vary.

For additional assistance in converting your designs, contact your Cadence customer support expert. They are there to help keep you working as fast, smart, and efficiently as possible.

Coming Soon…

The UPD package design product is being phased out. It is no longer for sale. Support of the product – including bug fixes – is scheduled to end at the end of the 2014 calendar year. Start converting your designs and gaining expertise with the APD and SiP Layout products to ensure you maintain the highest possible level of design efficiency.

Another advantage of migrating to the new products early is that, by learning the new tools now, you will be able to request specific functional enhancements to your new layout tools that will make your transition as seamless and pleasurable as possible.