

Virtual Reference Design from ROHM

PathWave ADS workspace for ROHM
P01SCT2080KE-EVK-001

Table of Contents

Dependencies / Pre-requisites from Keysight and ROHM.....	4
Installing Advanced Design System (ADS).....	5
Setting Up Licenses on Windows	6
Setting Up Floating Licenses on Windows.....	6
Setting Up Node-locked License on Windows	7
Dependencies / Pre-requisites from ROHM.....	10
Launching Advanced Design System (ADS)	10
Unarchiving the Advanced Design System (ADS) Workspace	12
Understanding the ADS Workspace	13
Running the Virtual Prototype “as-is”	14
PEPro Setup	16
Ports, Component Models, Nets, Options.....	17
Options Setup	20
Running a PEPro Simulation.....	22
Results	23
Design Space Exploration and “What if...?” Analysis	28
Conclusion.....	31

This application note provides information to obtain and use a PathWave ADS workspace that you can use to simulate a virtual version of a physical reference design from ROHM, Model P01SCT2080KE-EVK-001. This model showcases the second generation planar SiC power transistors (SCT2XXX series) also from ROHM. Provides the details on how they can be used to build your next-generation switched-mode power supply. In addition, the reference design – in both virtual and physical forms - can be used as starting point for your own project.



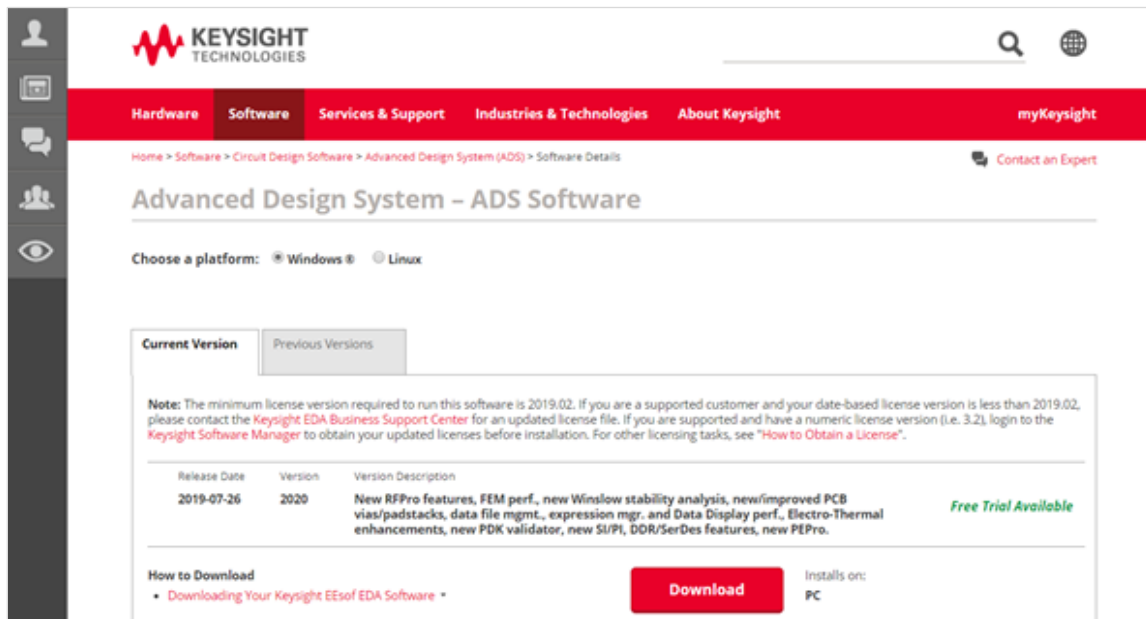
Dependencies / Pre-requisites from Keysight and ROHM

This section lists the dependencies / pre-requisites that you will require from Keysight and ROHM.

Dependencies / pre-requisites from Keysight

Step 1: Download and Install the PathWave ADS binaries

1. Go to <http://www.keysight.com/find/eesof-ads-latest-downloads>.
2. Download the ADS installation file(s):
64-bit: ads_20XX_win_x64.exe
to any folder, for example C : \Temp.



NOTE: If you need any assistance, click “Contact an Expert,” top right.

Step 2: Request a No-charge 45-days Evaluation License

1. Go to <http://www.keysight.com/find/eesof-ads-evaluation>

Step 3: Download the Workspace Archive for the ROHM Kit

The web link is <https://connectlp.keysight.com/Rohm-Evaluation-Platform>

The archived Workspace file name is ROHM_P01SCT2080KE_EVK_007-1.7zads

Installing Advanced Design System (ADS)

To install Advanced Design System on Windows, follow these steps.

1. Run the downloaded installation file from the location where you saved it. Before starting the installation, the installation file extracts all the necessary files in a temporary folder.
2. Select the language from the drop-down list of **Installation** window and click OK to continue.
3. On the **Introduction** screen, click **Next** to read the License Agreement. This is only a usage agreement and is not related to the licenses required to run the software.
4. On the **License Agreement** screen, accept the agreement and click Next.
5. On the **Choose Install Set** screen, choose any one of the two installation options:
 - **Complete:** For complete ADS installation. It requires about 4.5 GB of disk space and installs the ADS design environment, simulators, documentation, examples, and design guides. If you choose Complete installation, skip the next step.
 - **Custom:** For custom ADS installation where you can choose from the ADS items you want to install. While you can choose which ADS items you want to install, the ability to run them is determined by the purchased licenses.

6. For custom installation, the next screen lets you choose the items you want to install. Click any item name to see the description of the same in the window. Select the items you want to install and click **Next**.

You must choose to install ADS programs and tools which includes *Layout, Simulators and Schematic Capture*; else ADS will not run properly. Manuals and Help is selected by default to install the entire ADS documentation set. If you uncheck this item, documentation and help files will not be available regardless of which other items you choose.

7. At the Choose Install Folder screen, click:
 - **Next** to install to the default folder. The installation program creates this folder if it does not already exist. One more option **Restore Default Folder** is also provided to restore the default folder after every uninstallation.
 - **Choose** to specify a different destination folder. If you specify a folder that does not already exist, the installation program will create it for you.
The default folder for ADS installation is: **64-bit:** "C:\Program Files\Keysight\ADS20XX". You can change the default location and select your own desired path. Ensure that you must have permissions to write to the disk drive on which you want to install ADS. The installation also adds entries to your PC's Windows Registry, so ensure that you have permissions to do so.
8. At the Set Your Home Directory screen, click:
 - **Next** to accept the default.
 - **Choose** to select a different folder. The folder you specify will be the default startup folder for ADS. It will also be used to store the configuration data in a subfolder called `hpeesof`. For every single release and sub-release of ADS (such as ADS 2012.08, ADS 2013.06, ADS 2014.01, ADS 2015.01 and ADS 2016.XX), you must choose a new *Home* folder for each version. For the best performance, choose a folder on your local, not network drive.

9. At the Choose Shortcut Folder screen:
 - You can type a name in *Start Menu Folder Name* to have the same named folder in Start menu, by default it is Advanced Design Systems 2016.XX.
 - If you want ADS icon on desktop, check the *Icon To Desktop* option.
10. At the Pre-Installation Summary screen, review your choices, and click:
 - **Previous** to access previous screens to verify entries and make changes. This screen appears again if you go back to make other choices.
 - **Install** to start the installation. Immediately after you click **Next**, the *Setup Status* screen appears, and the program configures the installation process. When the program determines that your system is ready for installation, it begins to install files. The status screen displays the files being installed and the progress.
 - **Cancel** to exit the Setup without completing the installation.
11. After installation, click **Done** to quit the installer. The ADS License Setup Wizard starts automatically after this and check for the license status. Before launching ADS, you must complete the ADS license setup.

Setting Up Licenses on Windows

This section describes how to set up a node-locked license or a floating (network) license on Windows.

Both node-locked and floating licenses require license server to get installed and started. For a node-locked license, license server and the client machine are the same machine (that is, your local machine). For a floating license, these can be separate machines.

Setting Up Floating Licenses on Windows

Setting up floating licenses on Windows is a two-step process where both server-side setup and client-side setup are needed.

Step 1: License server setup

To setup a dedicated floating license server that has no Keysight EDA software installed, you will need to download and install the EEsof EDA License Server Software. This will install the license daemons and utilities needed to configure and start the license server.

Repeat the following steps for each license server you have.

Step 2: Client setup

Once the license is configured successfully for all the license servers, configure each client computer to point to the network license server. **License Setup Wizard** offers the easiest method to get this done. Alternatively, you can perform this step manually by following step 3 of Setting up Node-locked License on Windows section. Make sure to specify the network license server in the format of `port@host.domain` or if no static TCP port is used on the license server, use the format `@host.domain`.

Examples

`27001@myServer.myCompany.com`

`@myServer.myCompany.com`

In case a license server is down or is running slow, it is recommended to remove it from the server list using the **License Setup Wizard** to avoid any performance degradation caused by the dead / slow servers.

Setting Up Node-locked License on Windows

On Windows, the License Setup Wizard offers the easiest way to set up your node-locked license. We strongly recommend you to use the **License Setup Wizard**, especially when you have multiple Keysight EEsof products installed on one computer. For more information, refer to Quick Install-Windows. This section describes how to set up a node-locked license manually, should you decide not to use the **License Setup Wizard**.

Do not perform manual setup steps with the **License Setup Wizard**. Choose one method or the other. **License Setup Wizard** may not be able to replace what you set up manually due to Administrator Privilege differences.

The following instructions assume that you have already installed Keysight EEsof Licensing Tools in `C:\Program Files\Keysight\EEsof_License_Tools` folder. In case you have not yet installed the tools, run the Keysight EEsof Licensing Tools installer on the machine to which the license is tied to.

- If you have installed an EEsof product released before 1st August, 2014 on your system prior to installing an EEsof product released after 1st August, 2014 then the default EEsof Licensing tools path will remain `C:\Program Files\Agilent\EEsof_License_Tools`. On the systems that only have EEsof products released after 1st August, 2014 installed then the default EEsof Licensing tools path will be `C:\Program Files\Keysight\EEsof_License_Tools`.
- Make sure to save the license file on the same machine where you install the Keysight software.

The following 3-step process shows how to manually set up a counted license that is tied to a CPU ID.

Step 1: Start license server managers

Start the FlexNet license server managers (**lmgrd** and **agileesofd**) on the computer to which the license is tied as follows:

1. Open the MS-DOS command prompt and change directory to the following path:
C:\Program Files\Keysight\EEsof_License_Tools\bin.
2. Specify the full path and location of the license file and the license log file, respectively, by typing the following command:

lmgrd.exe -c <license file path> -l <license log file path>

where,

- <license file path> specifies one or more full pathnames to the license file(s).
- <license log file path> is the full pathname to the debug log file.

The spaces in pathnames requires double quotes around the path-names.

Example

```
lmgrd.exe -c "C:\Program  
Files\Keysight\EEsof_License_Tools\licenses\license.lic" -l  
"C:\Users\Public\my_server_log.txt"
```

Step 2: Create Windows service

To automate the restart of the license server managers upon PC reboot, configure a Windows Service as follows:

You must have Administrator Privileges to configure a Windows Service. The service will run under the Local System account.

1. Run the *lmtools.exe* from *C:\Program Files\Keysight\EEsof_License_Tools\bin*. The **lmtools** utility window is displayed.
2. In the **Service/License File** tab, select the **Configuration using Services** option.

3. Click the **Config Services** tab and enter the following details:
 - **Service Name:** Type the service name (for example, `EEsof EDA License Server`).
 - **Path to the `lmgrd.exe` file:** Click **Browse** and specify the path to the `lmgrd.exe` file on the license server (for example, `C:\Program Files\Keysight\EEsof_License_Tools\bin\lmgrd.exe`).
 - **Path to the license file:** Click **Browse** and specify the path to the license file on license server (for example, `C:\Program Files\Keysight\EEsof_License_Tools\licenses\license.lic`). If you have multiple license files for serving multiple Keysight EEsof products, you should include them all.
 - **Path to the debug log file:** Click **Browse** and specify the path to the debug log file on license server (for example, `C:\Users\Public\my_server_log.txt`).
4. Select the **Use Services** option.
5. Select the **Start Server at Power Up** option.
6. Click **Save Service**.
7. To verify that the service is created, click the **Start/Stop/Reread** tab and observe that your service shows up in the **FlexNet license services installed on this computer** list. To verify if the license managers are running, click again on the **Config Services** tab later and click **View Log**. A log window appears that confirms whether *lmgrd* and *agileesofd* are up and running.

Step 3: Set product environment variable

Set the environment variable `<PRODUCT>_LICENSE_FILE` to point the machine running the license server managers (*lmgrd* and *agileesofd*).

1. Select **Start > Control Panel > System**. Click **Advanced system settings**. The **System Properties** dialog box appears and the **Advanced** tab is selected by default).
2. Click **Environment Variables**.
3. In the **Environment Variables** dialog box, add the product specific environment variable `<PRODUCT>_LICENSE_FILE` as a new system variable. For example, for SystemVue, it is **SYSTEMVUE_LICENSE_FILE**, for ADS, it is **ADS_LICENSE_FILE**, for EMPro, it is **EMPRO_LICENSE_FILE** and so on.
4. In the **New System Variable** dialog box, set the value to be the same as what you specified in the `--c` argument when you start *lmgrd*. To use the example illustrated in Step 1, second bullet, this value would be `C:\Program Files\Keysight\EEsof_License_Tools\licenses\license.lic`.
5. Repeat 3 and 4 for each Keysight EEsof product you have installed on your computer.

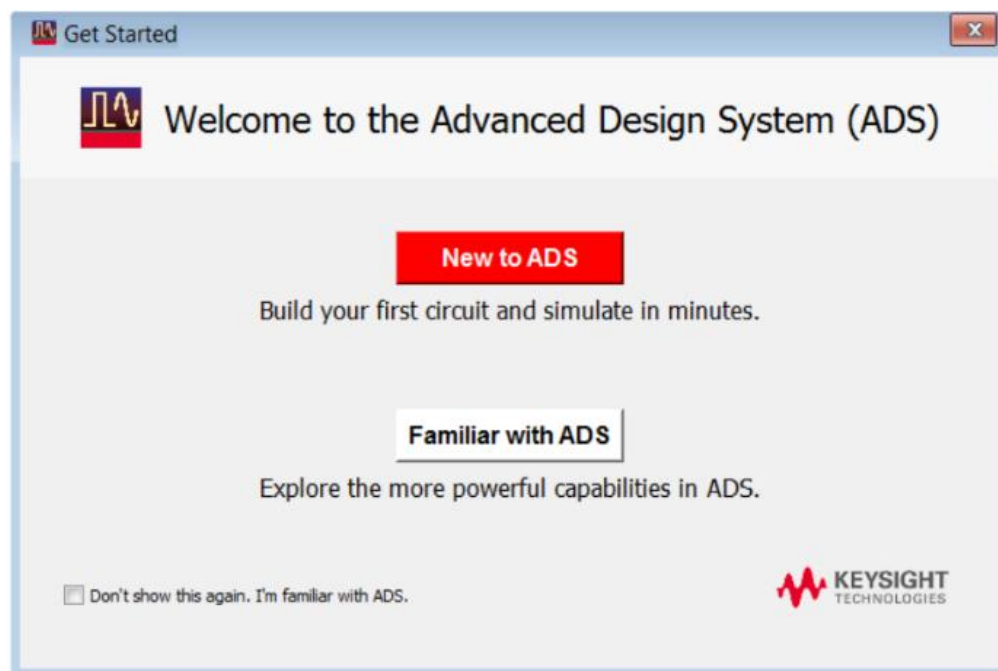
Dependencies / Pre-requisites from ROHM

For details on the ROHM hardware, including information on how obtain it, please visit <https://www.rohm.com/power-device-support>.

Launching Advanced Design System (ADS)

To start Advanced Design System (ADS), click **Start > Advanced Design System 201x.xx** from the taskbar.

- For existing ADS users, ADS starts normally.
- For a new ADS user, The Welcome to the Advanced Design System screen is displayed with two options:
 - **New to ADS** - Starts ADS Quick Start, which enables new users to discover the basics of ADS quickly.
 - **Familiar with ADS** - Starts ADS in normal mode and enables you to explore all the powerful and advanced ADS capabilities.



ADS quick start mode highlights

- **First-time User Detection:** ADS Quick Start mode is enabled by default for the first time ADS users. It can be enabled or disabled in the ADS (Main) preferences.
- **Simplified Experience:** Skips all the startup screens and wizards of the standard mode.
- **Overlay Tips:** Provides quick steps for getting started effectively with circuit drawing, setting up simulation, and plotting / viewing results.
- **Missing Step Detection:** Displays messages when critical steps have been missed such as placing a Simulation Controller or in some case a Source.

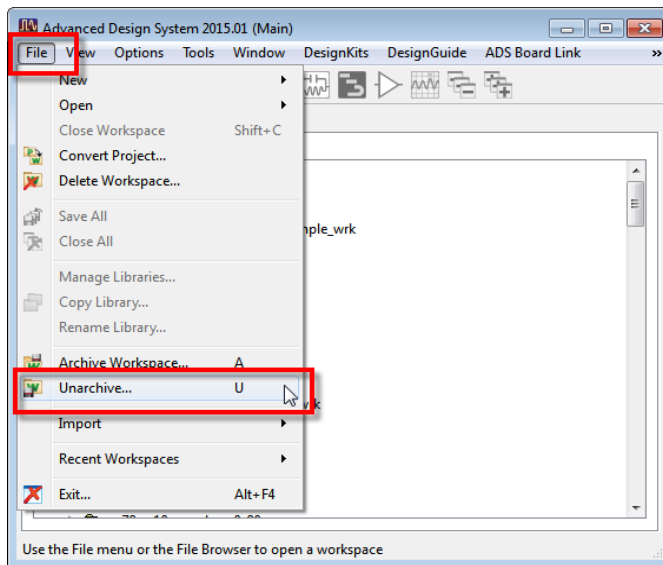
Tips to invoke ADS quick start mode

For new user	For existing user
When ADS starts, the Welcome to the Advanced Design System screen is presented. To turn off the screen:	ADS starts normally. To enable ADS quick start:
<ul style="list-style-type: none">• Check the "Don't show this again." on the lower left corner of the screen. or	<ul style="list-style-type: none">• Type ads -quickstart from the command prompt. or
<ul style="list-style-type: none">• From ADS Main window, click Options > Preferences and uncheck the "Enable Quickstart" checkbox.	<ul style="list-style-type: none">• Create a new ADS shortcut that calls ads - quickstart. Or• From ADS (Main) window, click Options > Preferences and check the "Enable Quickstart" checkbox

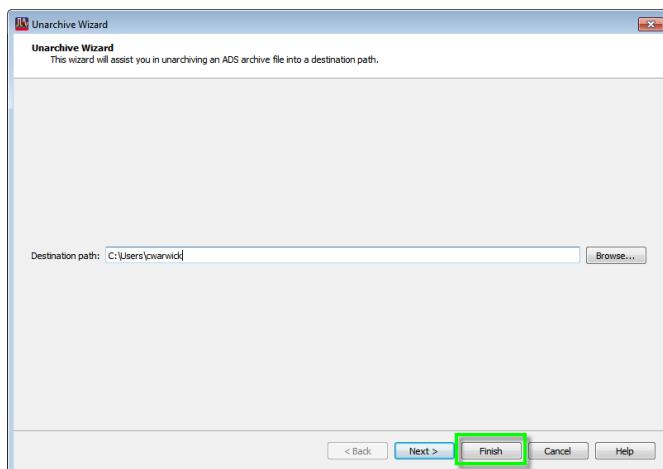
Unarchiving the Advanced Design System (ADS) Workspace

To unarchive the workspace:

1. Click **File > Unarchive**.
The Select an Archive File browse window is displayed.



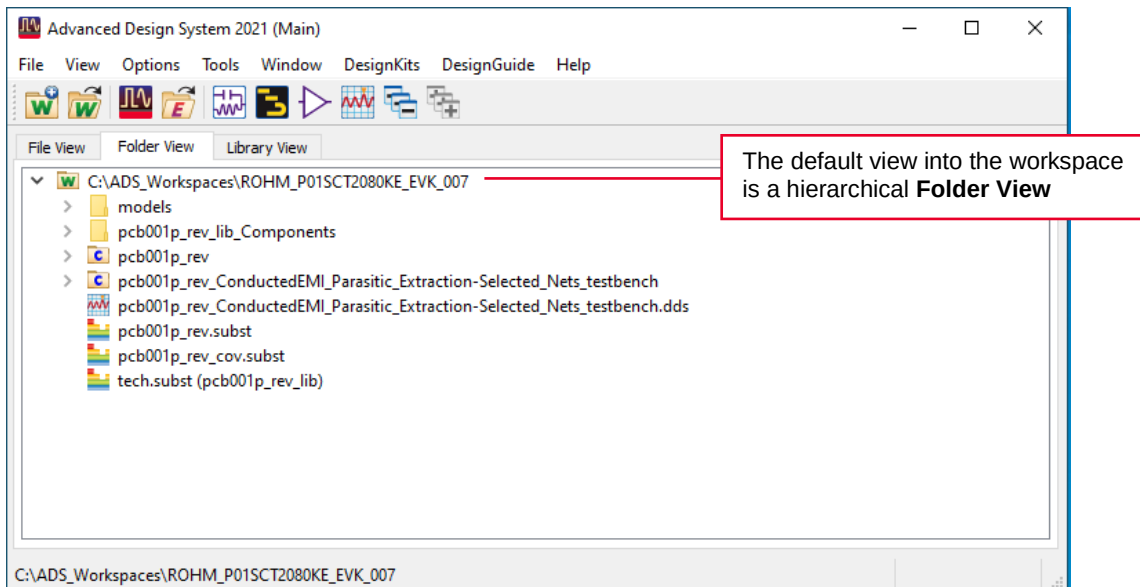
2. Browse to the folder where you downloaded the workspace.
3. Click **Open**.
The Unarchive wizard is displayed.



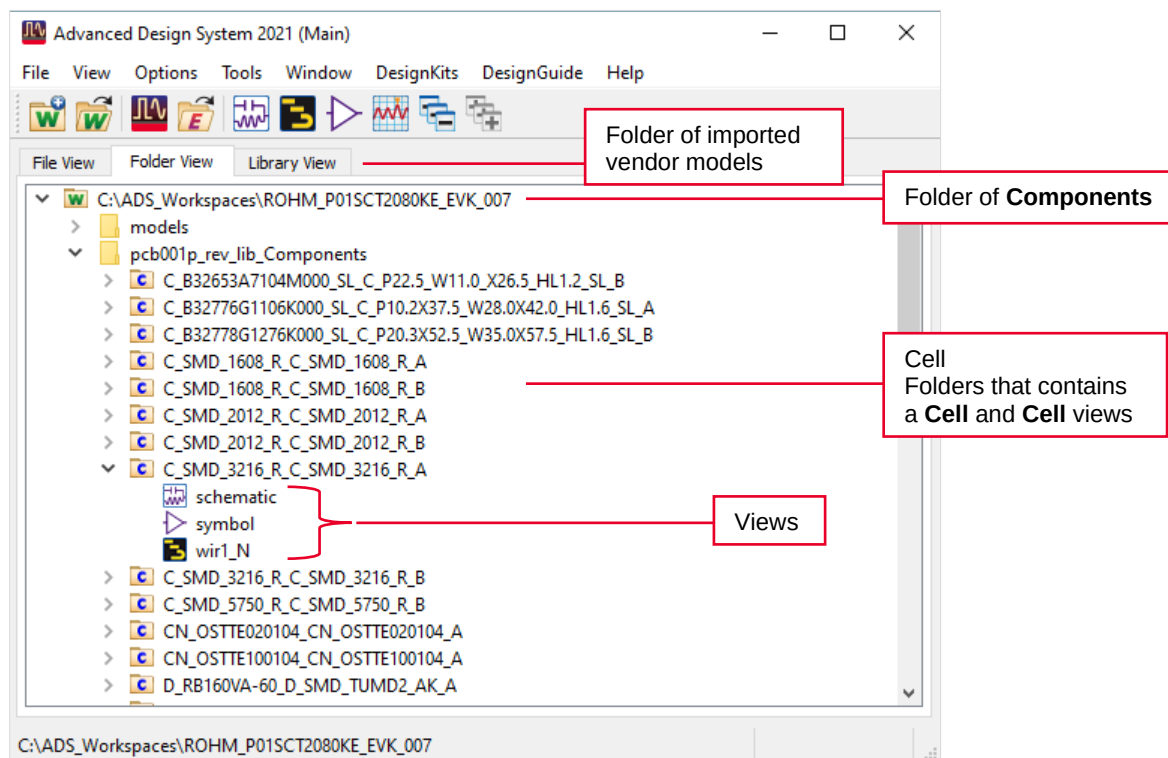
4. Browse to select the destination folder.
5. Click **Finish**.

Understanding the ADS Workspace

1. ADS main window



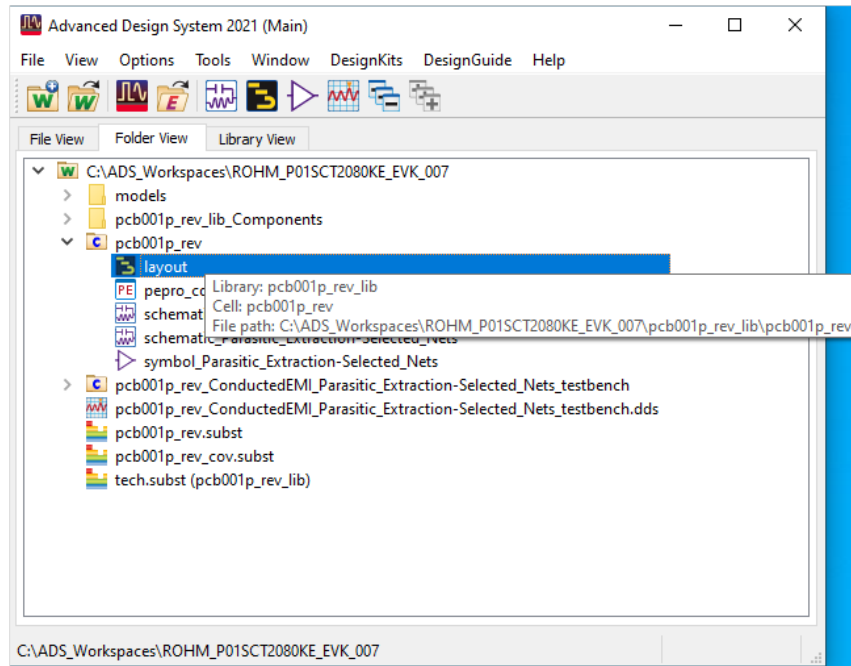
2. Folder of components



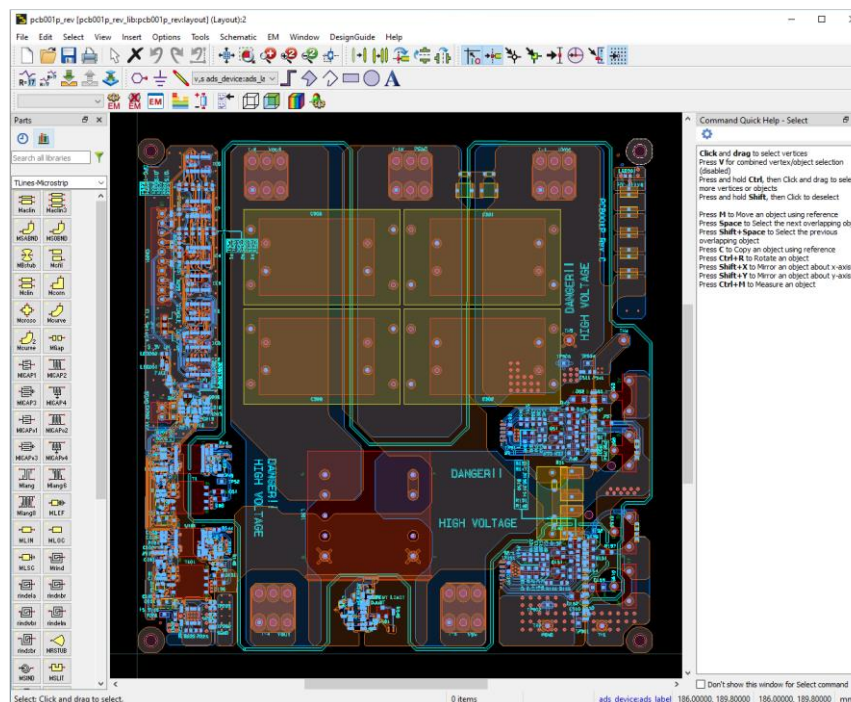
Running the Virtual Prototype “as-is”

ADS Layout View and Substrate Editor

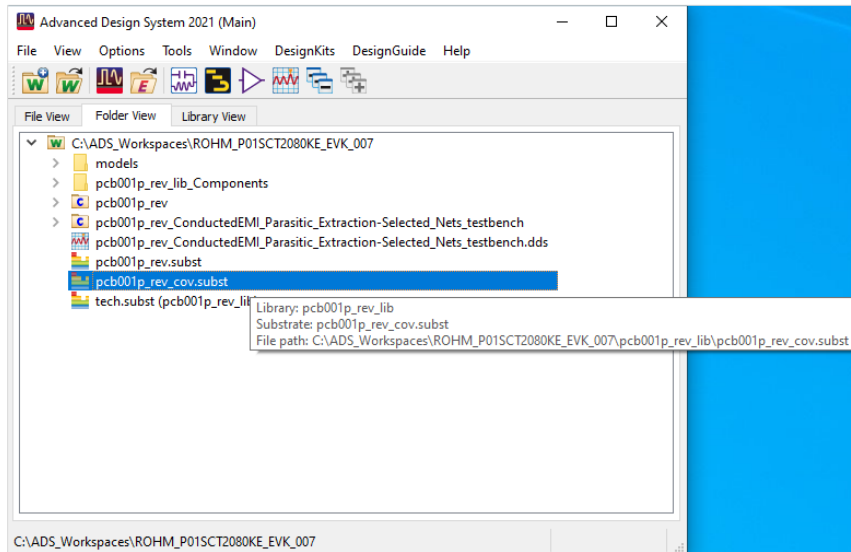
1. Double-click layout view of pcb001p_rev.



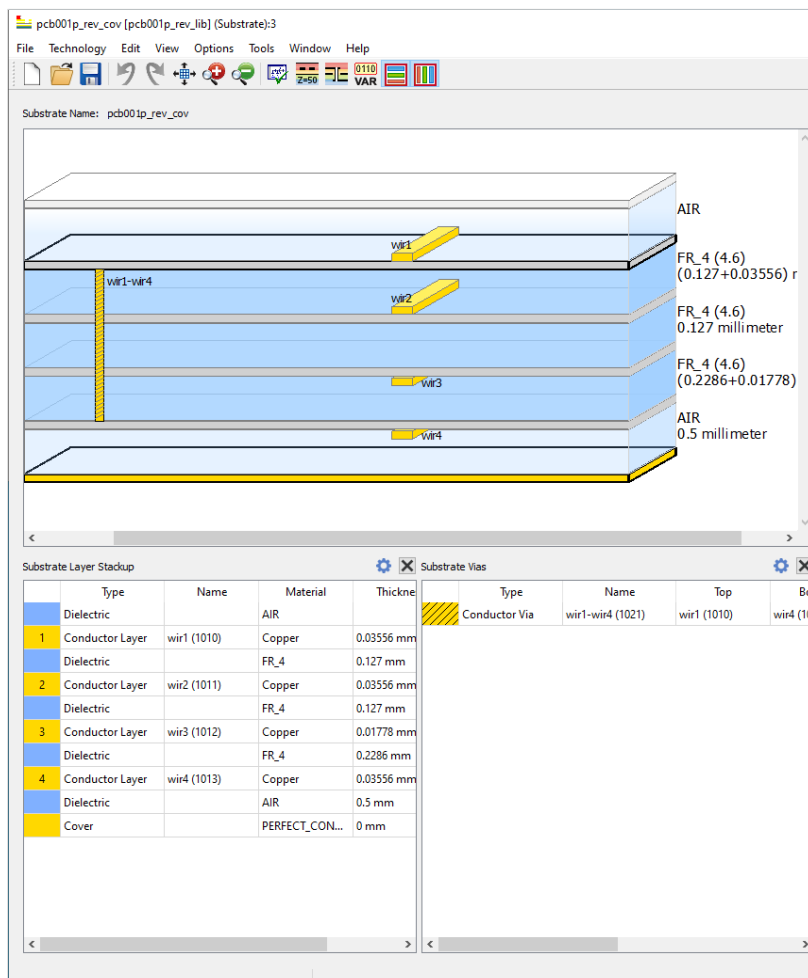
This view provides a possible layout with all the circuit components for this design. It can then be opened in **PEPro** where an **EM** model extraction can be performed.



2. To view/edit the substrate, double-click the substrate in the ADS main window.

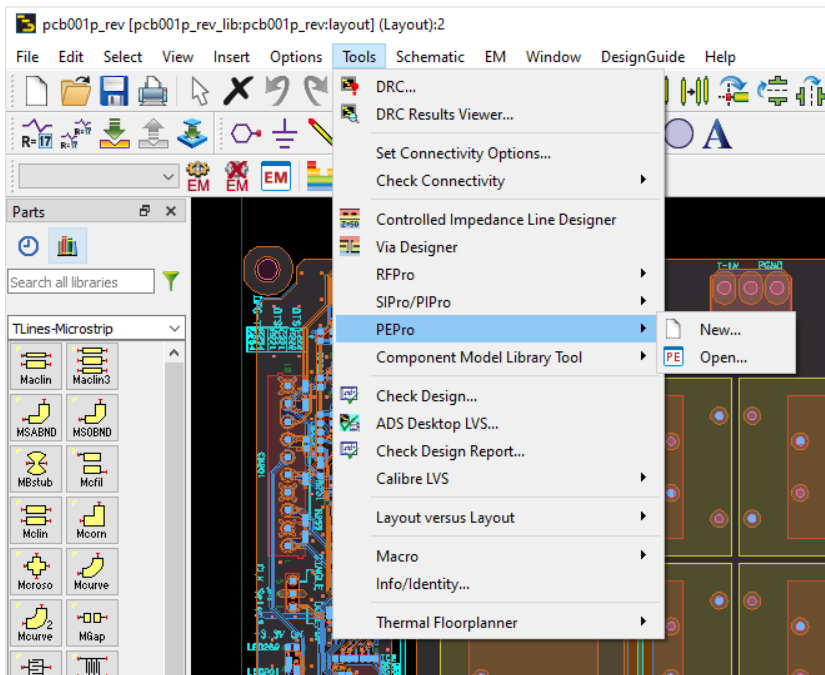


The Substrate Editor window view:

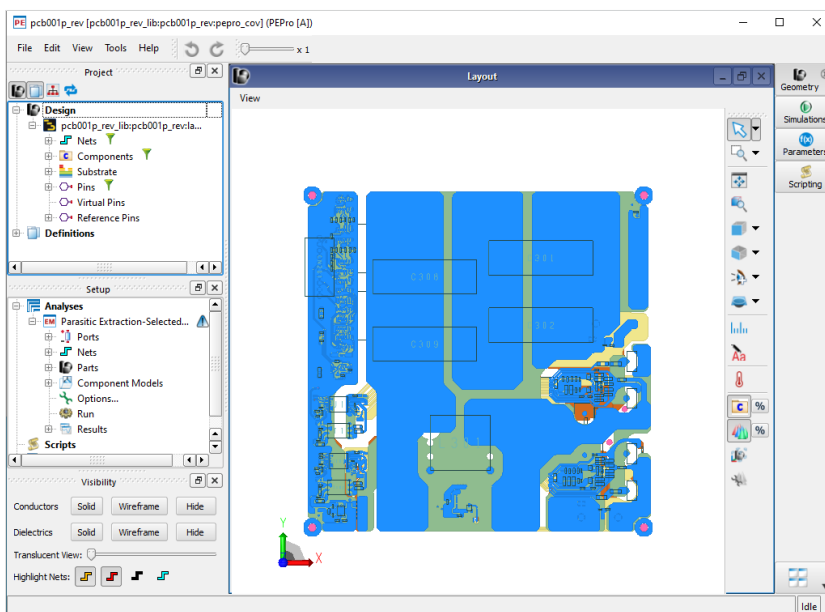


PEPro Setup

1. In the Layout window, click **Tools > PEPro > Open**.



2. In the PEPro Main window, view the Project and Setup dockable sub-windows content. The Project sub-window contains all elements included in the layout that is nets, components, pins, and so on. The Setup sub-window is used for EM analysis of either entire design (Parasitic Extraction-All Nets) or only selected nets (Parasitic Extraction-Selected Nets).

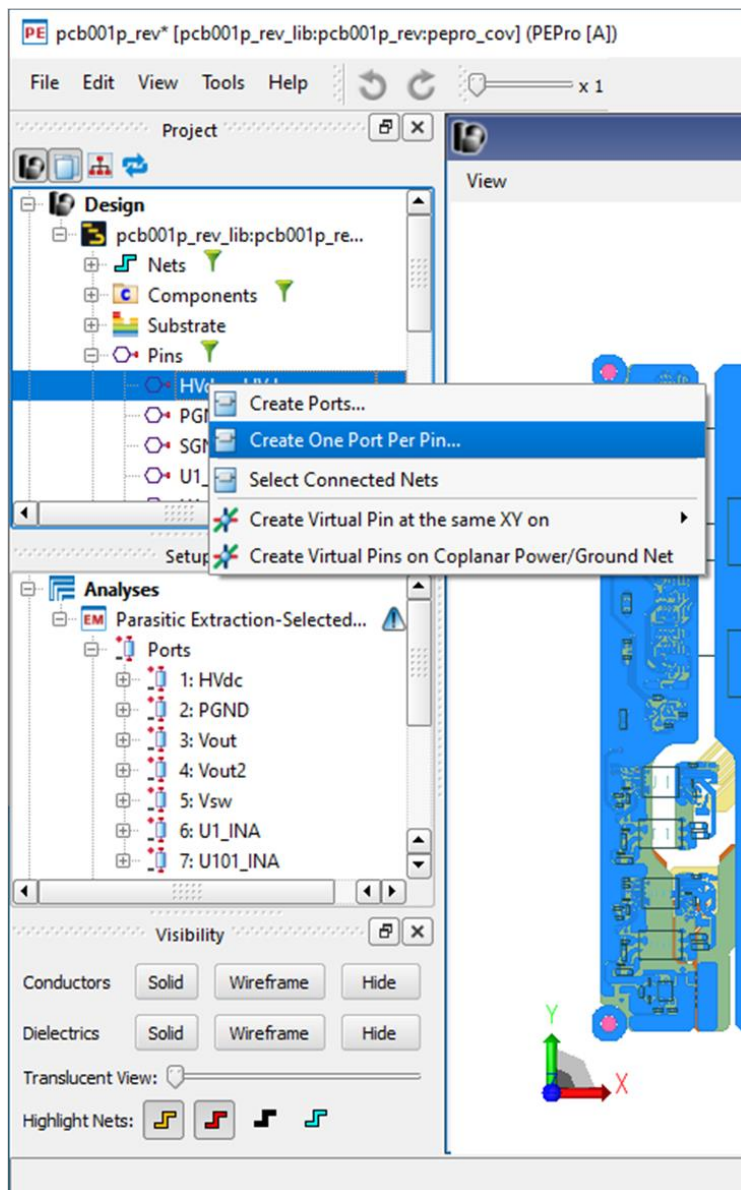


Ports, Component Models, Nets, Options

In the example, Ports, Component Models, Nets, Options have already been configured. You can double-click the **Run** button to run an EM Model Extraction. To understand the setup, read the details in the subsequent sections.

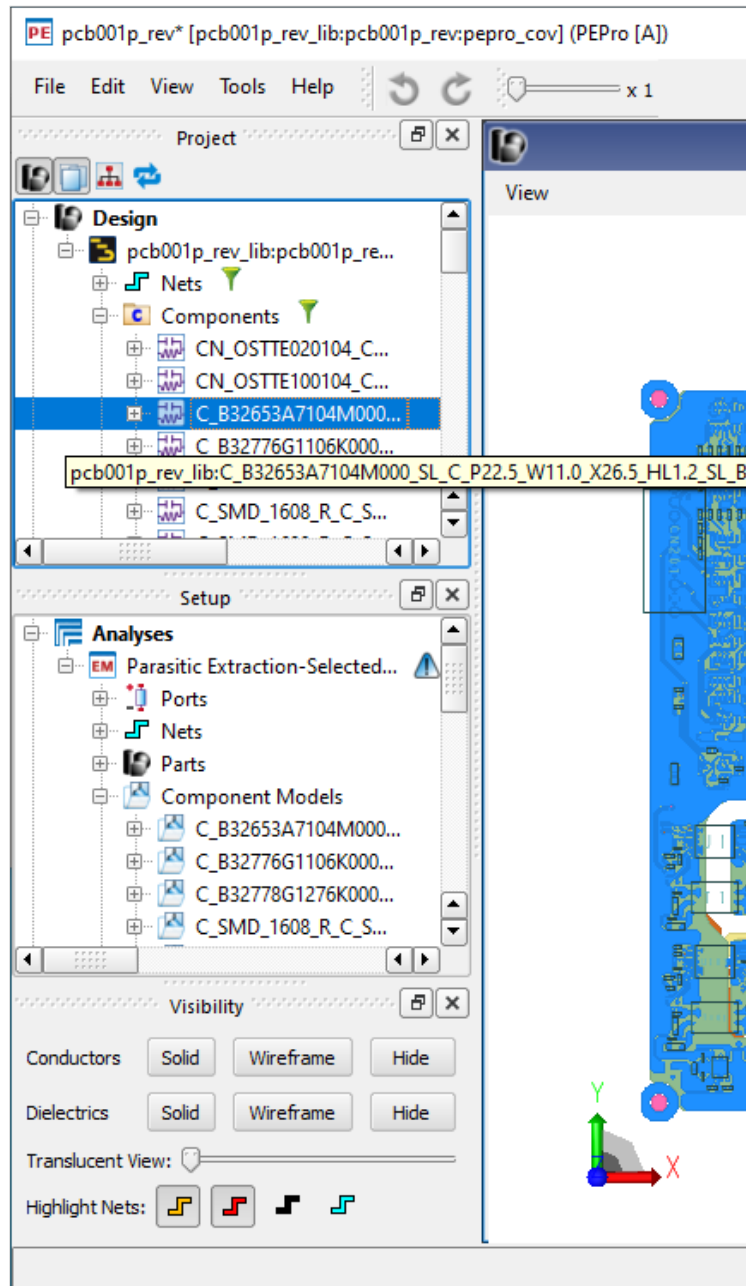
Ports setup

Select pins under **Design > Pins**. Right-click and chose **Create One Port per Pin**.



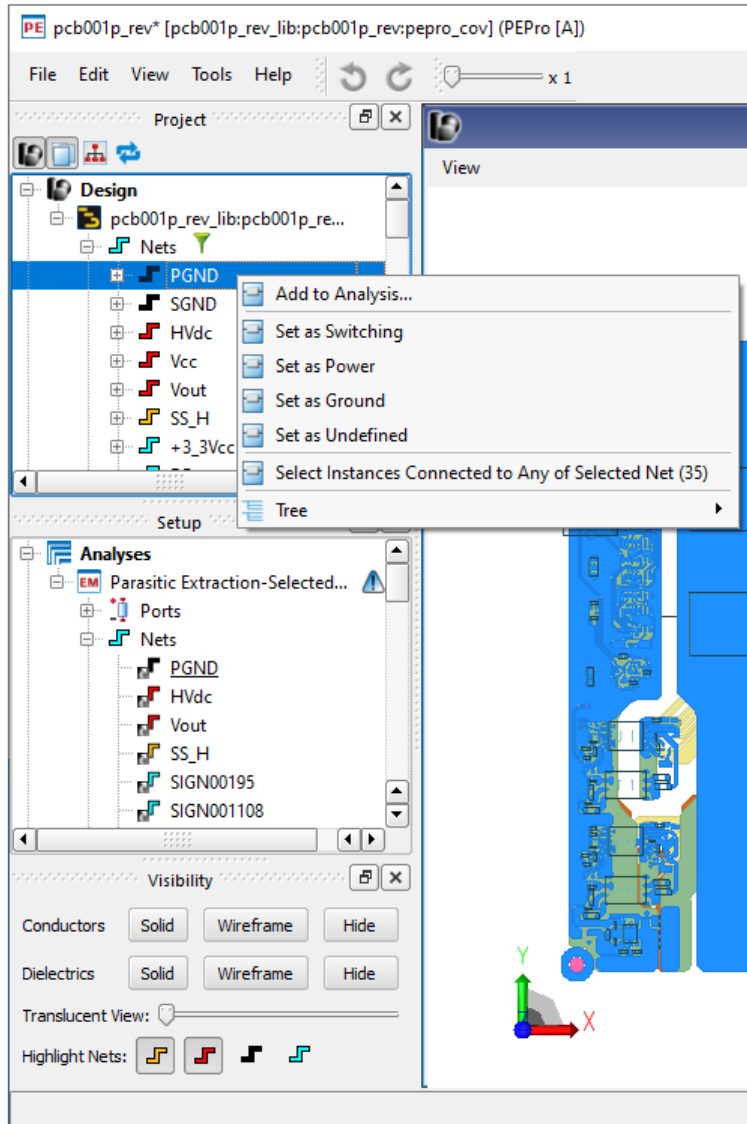
Components setup

Select the component instances under **Design > Components**. Drag them to the **Setup** sub-window and drop onto **Analyses**.



Nets setup

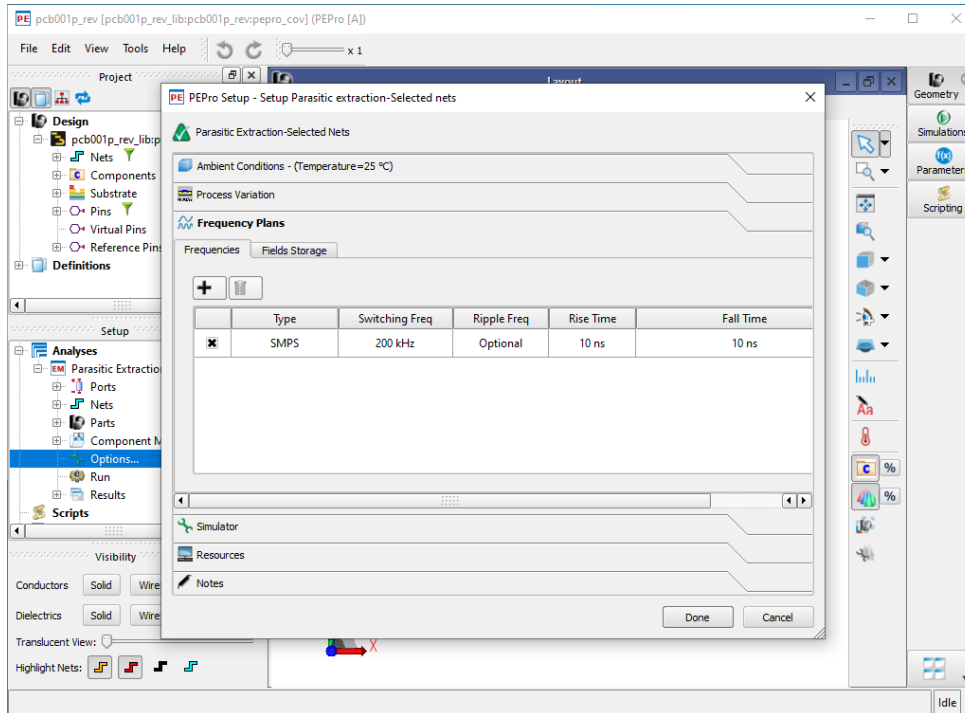
Select the nets under **Design > Nets**. Right-click and set them as Switching, Power, Ground or Undefined. Drag the selected nets to the **Setup** sub-window and drop onto the **Analyses**.



Options Setup

Frequency plans

There are several types of frequency sweeps. For switched mode power converters, we use an automatic frequency plan for switched-mode power supplies (**SMPS**) in Power Electronics. It creates a composite frequency plan that can capture the most important EM features of a PCB in SMPS circuits. To use this plan, the user needs to provide circuit characteristics in the frequency plan table. Once the required input data is set, the frequency plan will be generated automatically and used in the subsequent EM simulation.

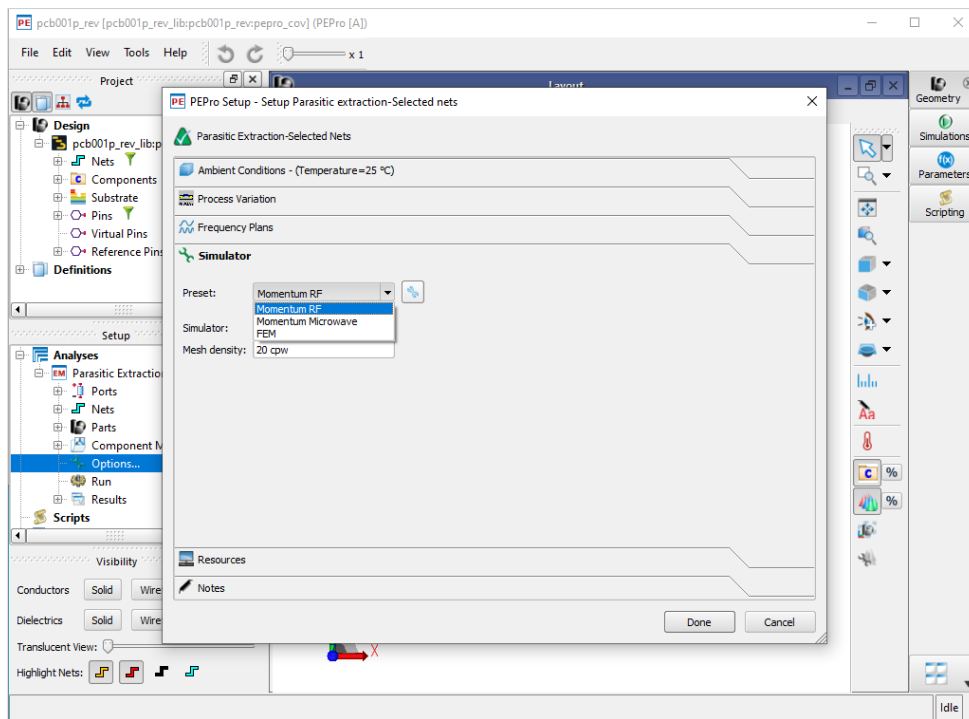


Simulator

There are three **Simulator Presets** available:

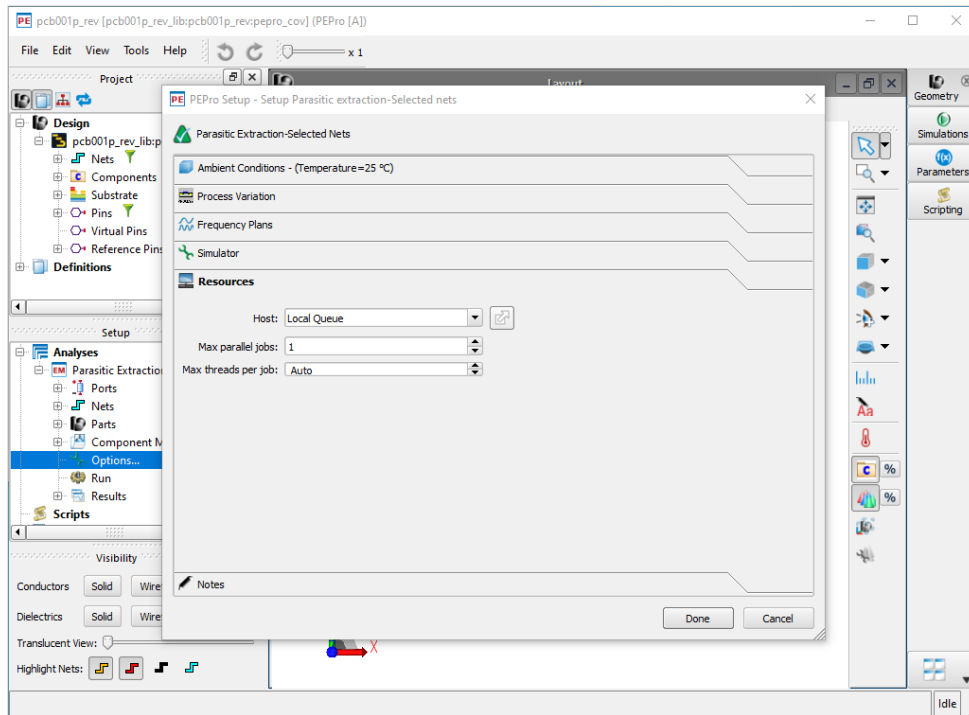
- Momentum RF
- Momentum Microwave
- FEM

Momentum RF provides the fastest EM simulation speed with good accuracy. Momentum RF is a fast, quasi-static 3D planar solver that is appropriate for SMPS PCBs and packages. The slower Momentum Microwave setting is intended for a different purpose, namely multi-gigahertz microwave circuits where the highest wavelength, that is, speed of light divided by highest frequency is physically much smaller than the size of the board. FEM is intended for objects with arbitrary 3D geometries such as connectors or boards with very high layer count.



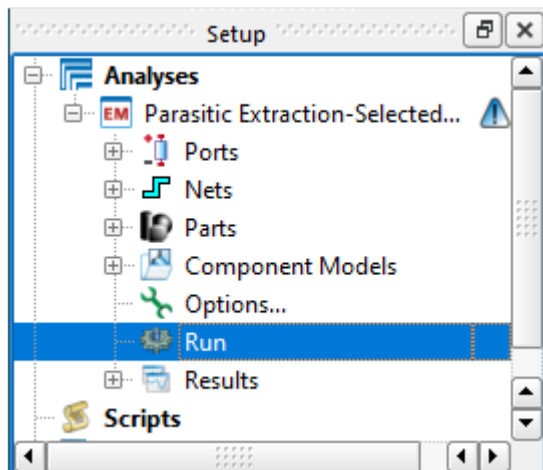
Resources

You can run EM simulations on a local or remote host machine and benefit from faster simulation times.



Running a PEPro Simulation

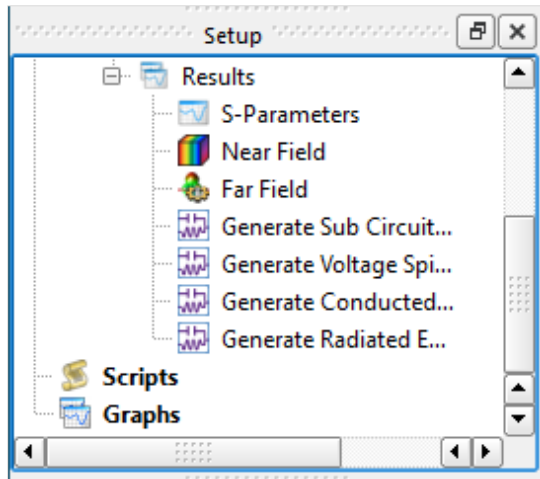
Double-click **Run** to start the EM simulation. The simulation takes several minutes depending on available computation resources. When the simulation completes, close the **Simulations** window.



Results

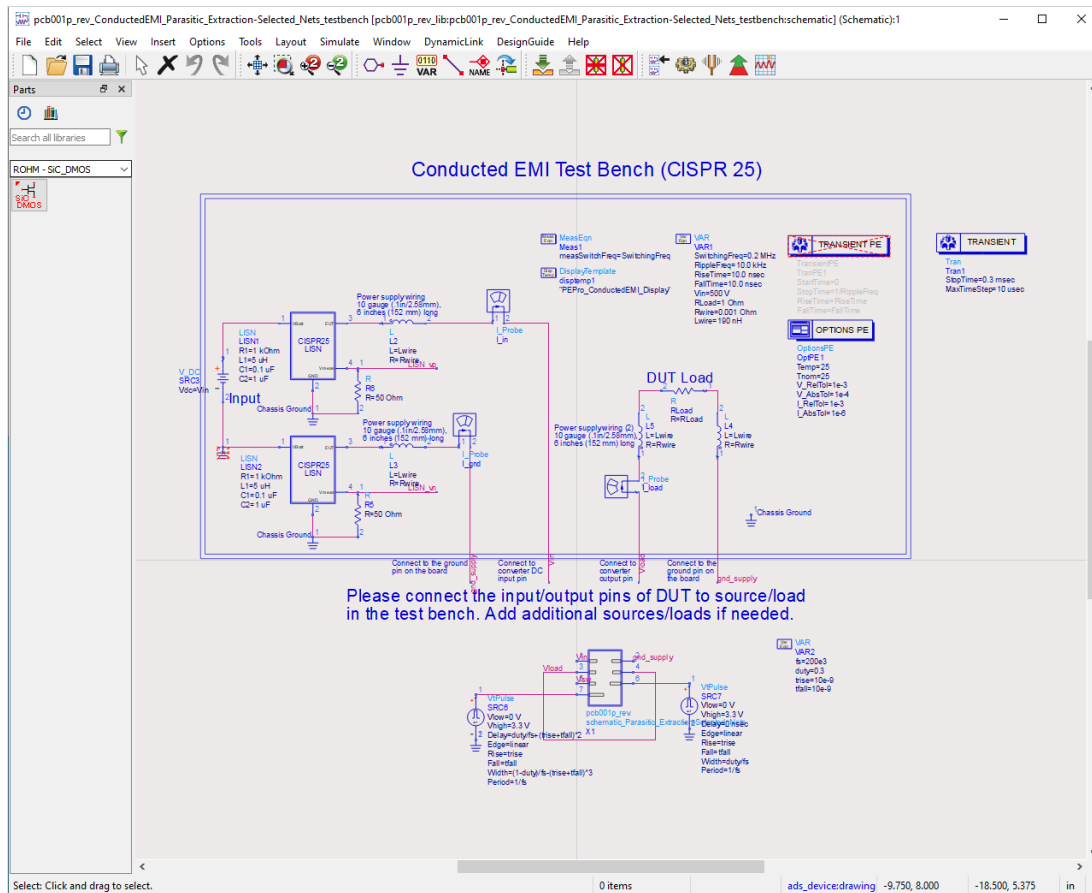
Generate sub circuit

Once the simulation is complete, double click **Generate Sub Circuit** to generate an EM model that can be used in the EM-circuit co-simulation. If the sub circuit already exists in the workspace, it may overwrite with the new EM simulation data. The example already contains a sub-circuit named **schematic_Parasitic_Extraction-Selected_Nets** and it will be overwritten (updated).



Generate testbench

Testbenches for Voltage Spikes, Conducted EMI and Radiated EMI are available. The example already contains a testbench named **pcb001p_rev_ConductedEMI_Parasitic_Extraction-Selected_Nets_testbench**. The testbench can be accessed from the ADS main window.

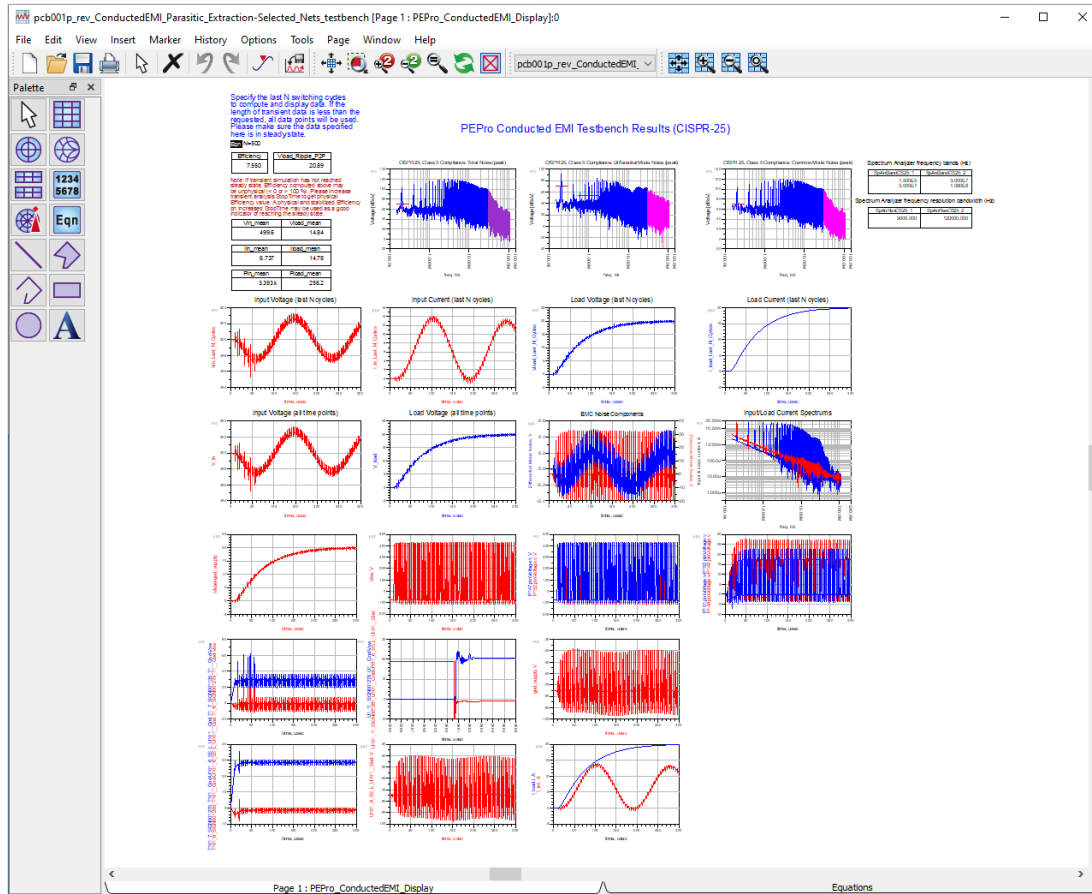


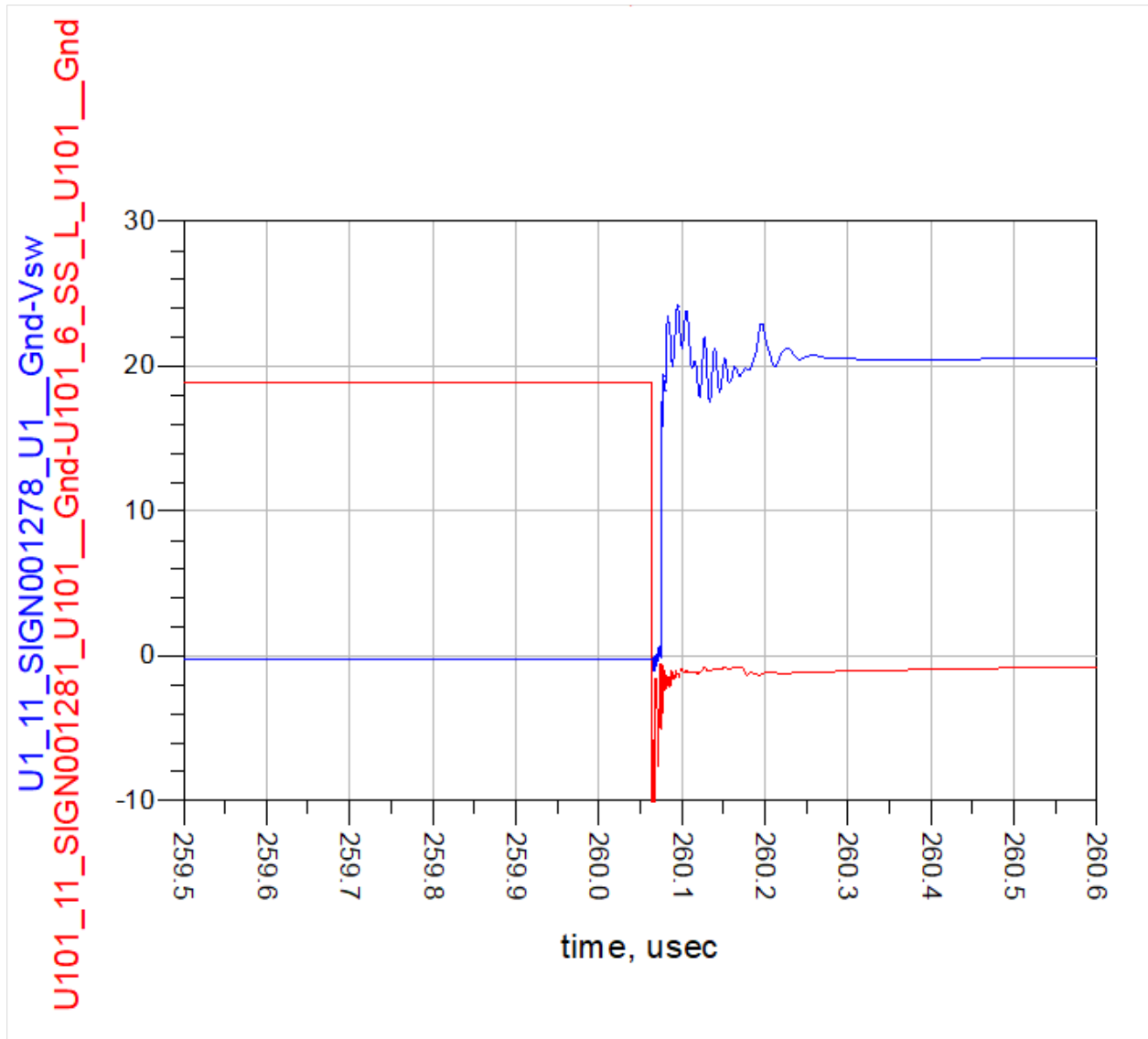
Simulate testbench

Make the required connections in the testbench. The testbench in the example already has the connections. To run simulation, select **Simulate > Simulate** or use the F7 function key.

View simulation results

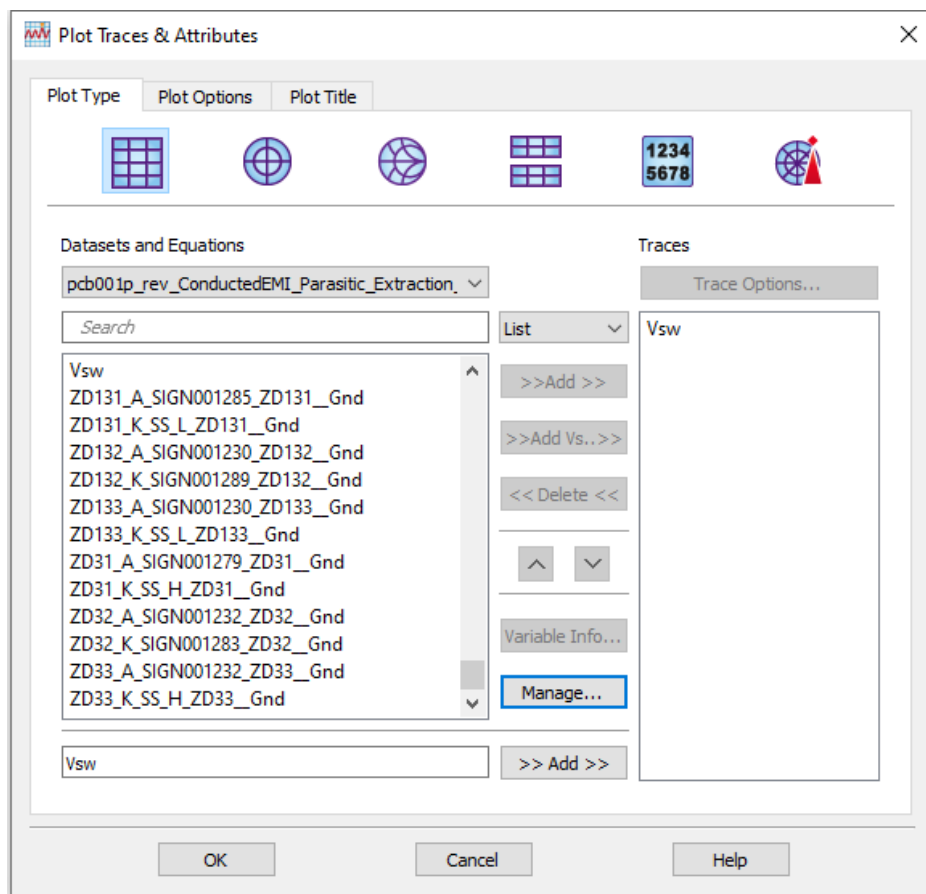
When the simulation is complete, a data display window is displayed with the results. The example uses a data display template on which the simulation results are plotted.





Create a new data display window

1. In the Data Display Window, click **Page > New Page** and accept the default name.
2. Select the Rectangular Plot icon under Palette window and place empty frame on Data Display Window's area. The Plot Trace and Attributes window is displayed.
3. In the left-bottom, enter any Equation field type equation and press the **Add** button next to this entry field. The equation should appear in the right window.
4. When task is completed, press **OK**.



Current density and radiated EMI results

Depending on the simulation settings in PEPro, you may be able to access Current Density and Radiated EMI results from the PEPro Results section.

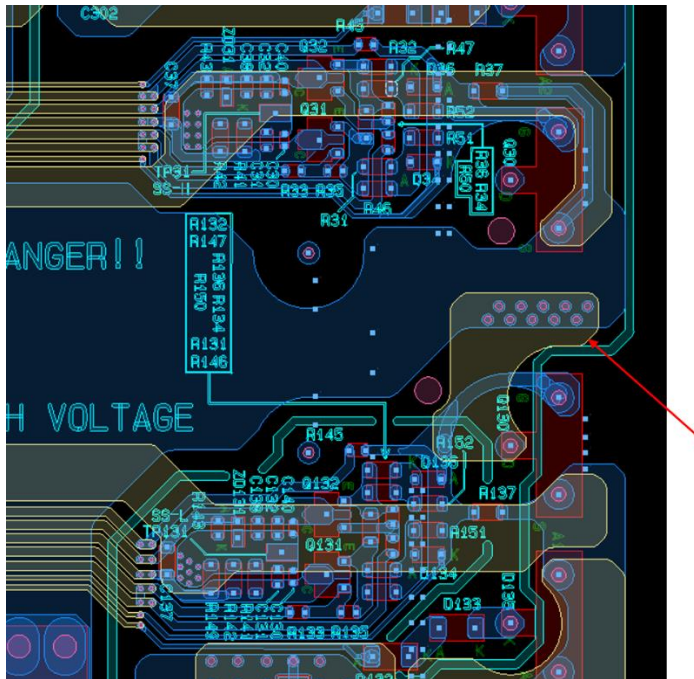
Design Space Exploration and “What if...?” Analysis

In contrast to a sequence of expensive and time consuming “solder-and-see” physical prototypes, the virtual prototype is more cost-effective for exploring the design space and performing “What if...” analysis. For example, the reference design as-is may be close to what you need, but not an exact match. But if you modify it you run the risk of introducing unwanted side effects like excessive voltage spikes and EMI problems. The virtual prototype gives you insight on how to change it while minimizing side effects. Try replacing some of the components with lower cost, lower performance ones. An example would a capacitor with higher series inductance or resistance and hence a lower self-resonant frequency. Next try re-routing the PCB traces, especially those in the switched loop that has high di/dt trapezoidal waveforms. See how the ringing and EMI are affected.

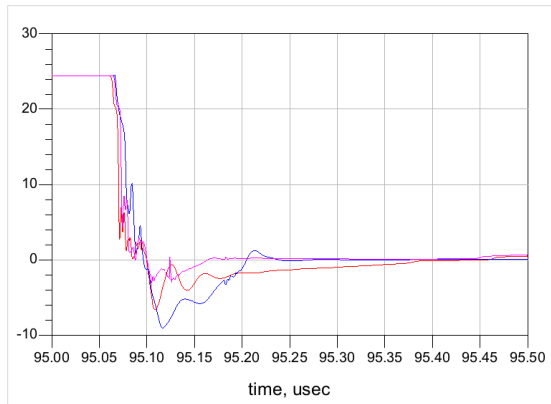
Example: Re-routing project

Consider the following project:

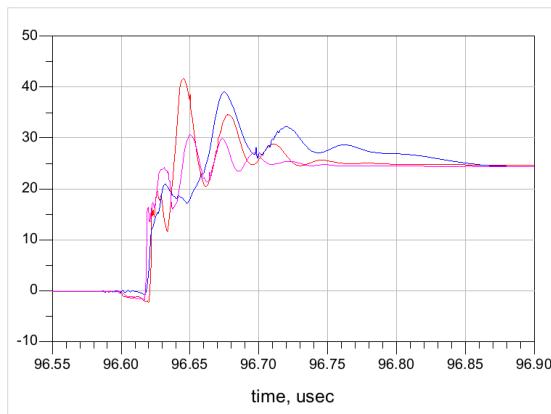
Your task is to clear a lane for additional circuitry on the layer marked in yellow here:



The result of the both modification is shown in pink, along with the original (red) and the removal-only modification (blue):



Turn-off waveform at the gate.



Turn on waveform at the gate.

The blue waveforms are unacceptable (undershoot and turn on time), but the red and pink are acceptable. In addition, the configuration that yields the pink waveform gives us the space we need.

Conclusion

Virtual prototypes like this one are complementary to physical prototypes. Physical prototypes are the gold standard for compliance and measured characteristics of course but they have several drawbacks:

- Expensive and time consuming to design, build, and measure
- Vulnerable to catastrophic failure – the infamous smoke test that produces actual smoke
- Hard to get a measurement probe onto interior nodes so you can't really see what is going on inside

In contrast, virtual prototypes are easy to change and while they do indicate device overstress, they never emit real smoke. The voltage, current, and fields at every spatial point in the 3D grid and every time step in the simulation can be recorded and made accessible for plotting, even point that are physical inaccessible in the real world such as inside a semiconductor package.



Keysight enables innovators to push the boundaries of engineering by quickly solving design, emulation, and test challenges to create the best product experiences. Start your innovation journey at www.keysight.com.

This information is subject to change without notice. © Keysight Technologies, 2020 – 2023, Published in USA, July 24, 2023, 3120-1476.EN