ADS
2016.01

SIPro and PIPro

KEYSIGHT
TECHNOLOGIES

Notice

Acknowledgments
Layout Boolean Engine by Klaas Holwerda, v1.7 http://boolean.klaasholwerda.nl/bool.html

FreeType Project, Copyright (c) 1996-1999 by David Turner, Robert Wilhelm, and Werner Lemberg.

QuestAgent search engine (c) 2000-2002, JObjects.

SuperLU_MT version 2.0

7-zip

Licenses for files are:

1. 7z.dll: GNU LGPL + unRAR restriction.
2. All other files: GNU LGPL.

7-zip Availability: http://www.7-zip.org/

AMD Version 2.2

AMD Notice: The AMD code was modified. Used by permission.

Errata

The ADS product may contain references to "HP" or "HPEESOF" such as in file names and directory names. The business entity formerly known as "HP EEsof" is now part of Keysight Technologies and is known as "Keysight EEsof". To avoid broken functionality and to maintain backward compatibility for our customers, we did not change all the names and labels that contain "HP" or "HPEESOF" references.

Qt Version 4.8.4

Qt Notice: The Qt code was modified. Used by permission.

Qt Availability: http://www.qtsoftware.com/downloads

Patches Applied to Qt can be found in the installation at: $HPEESOF_DIR/prod/licenses/thirdparty/qt/patches.

You may also contact Brian Buchanan at Keysight Inc. at brian_buchanan@keysight.com for more information.

The HiSIM_HV source code, and all copyrights, trade secrets or other intellectual property rights in and to the source code, is owned by Hiroshima University and/or STARC.

HDF5

HDF5 Notice: The HDF5 code was modified. Used by permission.

HDF5 License:

Copyright Notice and License Terms for HDF5 (Hierarchical Data Format 5) Software Library and Utilities

----

HDF5 (Hierarchical Data Format 5) Software Library and Utilities

libpng

libpng versions 0.97, January 1998, through 1.0.6, March 20, 2000, are Copyright (c) 1998, 1999 Glenn Randers-Pehrson, and are distributed according to the same disclaimer and license as libpng-0.96, with the following individuals added to the list of Contributing Authors: Tom Lane, Glenn Randers-Pehrson, Willem van Schaik

libpng versions 0.89, June 1996, through 0.96, May 1997, are Copyright (c) 1996, 1997 Andreas Dilger Distributed according to the same disclaimer and license as libpng-0.88, with the following individuals added to the list of Contributing Authors: John Bowler, Kevin Bracey, Sam Bushell, Magnus Holmgren, Greg Roelofs, Tom Tanner

libpng versions 0.5, May 1995, through 0.88, January 1996, are Copyright (c) 1995, 1996 Guy Eric Schalnat, Group 42, Inc.
For the purposes of this copyright and license, "Contributing Authors" is defined as the following set of individuals: Andreas Dilger, Dave Martindale, Guy Eric Schalnat, Paul Schmidt, Tim Wegner

The PNG Reference Library is supplied "AS IS". The Contributing Authors and Group 42, Inc. disclaim all warranties, expressed or implied, including, without limitation, the warranties of merchantability and of fitness for any purpose. The Contributing Authors and Group 42, Inc. assume no liability for direct, indirect, incidental, special, exemplary, or consequential damages, which may result from the use of the PNG Reference Library, even if advised of the possibility of such damage.
Permission is hereby granted to use, copy, modify, and distribute this source code, or portions hereof, for any purpose, without fee, subject to the following restrictions:

1. The origin of this source code must not be misrepresented.
2. Altered versions must be plainly marked as such and must not be misrepresented as being the original source.
3. This Copyright notice may not be removed or altered from any source or altered source distribution.

The Contributing Authors and Group 42, Inc. specifically permit, without fee, and encourage the use of this source code as a component to supporting the PNG file format in commercial products. If you use this source code in a product, acknowledgment is not required but would be appreciated.

OpenSSL

The OpenSSL toolkit stays under a dual license, i.e. both the conditions of the OpenSSL License and the original SSLeay license apply to the toolkit. See below for the actual license texts. Actually both licenses are BSD-style Open Source licenses. In case of any license issues related to OpenSSL please contact openssl-core@openssl.org.

OpenSSL License

==

Copyright (c) 1998-2011 The OpenSSL Project. All rights reserved.

Redistribution and use in source and binary forms, with or without modification, are permitted provided that the following conditions are met:

1. Redistributions of source code must retain the above copyright notice, this list of conditions and the following disclaimer.
2. Redistributions in binary form must reproduce the above copyright notice, this list of conditions and the following disclaimer in the documentation and/or other materials provided with the distribution.

Original SSLeay License

3. All advertising materials mentioning features or use of this software must display the following acknowledgement: "This product includes cryptographic software written by Eric Young (eay@cryptsoft.com)" The word 'cryptographic' can be left out if the rouines from the library being used are not cryptographic related.
4. If you include any Windows specific code (or a derivative thereof) from the apps directory (application code) you must include an acknowledgement: "This product includes software written by Tim Hudson (tjh@cryptsoft.com)"

THIS SOFTWARE IS PROVIDED BY ERIC YOUNG ``AS IS'' AND ANY EXPRESS OR IMPLIED WARRANTIES, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE ARE DISCLAIMED. IN NO EVENT SHALL THE AUTHOR OR CONTRIBUTORS BE LIABLE FOR ANY DIRECT, INDIRECT, INCIDENTAL, SPECIAL, EXEMPLARY, OR CONSEQUENTIAL DAMAGES (INCLUDING, BUT NOT LIMITED TO, PROCUREMENT OF SUBSTITUTE GOODS OR SERVICES; LOSS OF USE, DATA, OR PROFITS; OR BUSINESS INTERRUPTION) HOWEVER CAUSED AND ON ANY THEORY OF LIABILITY, WHETHER IN CONTRACT, STRICT LIABILITY, OR TORT (INCLUDING NEGLIGENCE OR OTHERWISE) ARISING IN ANY WAY OUT OF THE USE OF THIS SOFTWARE, EVEN IF ADVISED OF THE POSSIBILITY OF SUCH DAMAGE.

The licence and distribution terms for any publically available version or derivative of this code cannot be changed. i.e. this code cannot simply be copied and put under another distribution license \[including the GNU Public Licence.]

Growl GNTP support:
[The "BSD licence"] Copyright (c) 2009-2010 Yasuhiro Matsumoto
All rights reserved.

Redistribution and use in source and binary forms, with or without modification, are permitted provided that the following conditions are met:

1. Redistributions of source code must retain the above copyright notice, this list of conditions and the following disclaimer.
2. Redistributions in binary form must reproduce the above copyright notice, this list of conditions and the following disclaimer in the documentation and/or other materials provided with the distribution.
3. The name of the author may not be used to endorse or promote products derived from this software without specific prior written permission.

THIS SOFTWARE IS PROVIDED BY THE AUTHOR "AS IS'' AND ANY EXPRESS OR IMPLIED WARRANTIES, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE ARE DISCLAIMED. IN NO EVENT SHALL THE AUTHOR BE LIABLE FOR ANY DIRECT, INDIRECT, INCIDENTAL, SPECIAL, EXEMPLARY, OR CONSEQUENTIAL DAMAGES (INCLUDING, BUT NOT LIMITED TO, PROCUREMENT OF SUBSTITUTE GOODS OR SERVICES; LOSS OF USE, DATA, OR PROFITS; OR BUSINESS INTERRUPTION) HOWEVER CAUSED AND ON ANY THEORY OF LIABILITY, WHETHER IN CONTRACT, STRICT LIABILITY, OR TORT (INCLUDING NEGLIGENCE OR OTHERWISE) ARISING IN ANY WAY OUT OF THE USE OF THIS SOFTWARE, EVEN IF ADVISED OF THE POSSIBILITY OF SUCH DAMAGE.

Cuda
Cuda Redistributable Software - 1.8. Attachment A

In connection with Section 1.2.1.1 of this Agreement, the following files may be redistributed with software applications developed by Licensee, including certain variations of these files that have version number or architecture specific information embedded in the file name - as an example only, for release version 6.0 of the 64-bit Windows software, the file cudart64_60.dll is redistributable.

Component : CUDA Runtime
Windows : cudart.dll, cudart_static.lib
MacOS : libcudart.dylib, libcudart_static.a
Linux : libcudart.so, libcudart_static.a
Android : libcudart.so, libcudart_static.a

Component : CUDA FFT Library
Windows : cufft.dll
MacOS : libcufft.dylib
Linux : libcufft.so
Android : libcufft.so

Component : CUDA BLAS Library
Windows : cublas.dll
MacOS : libcublas.dylib
Linux : libcublas.so
Android : libcublas.so

Component : CUDA Sparse Matrix Library
Windows : cusparse.dll
MacOs : libcusparse.dylib
Linux : libcusparse.so
Android : libcusparse.so

Component : CUDA Random Number Generation Library
Windows : curand.dll
MacOs : libcurand.dylib
Linux : libcurand.so
Android : libcurand.so

Component : NVIDIA Performance Primitives Library
Windows : nppc.dll, nppi.dll, npps.dll
MacOs : libnppc.dylib, libnppi.dylib, libnpps.dylib
Linux : libnppc.so, libnppi.so, libnpps.so
Android : libnppc.so, libnppi.so, libnpps.so

Component : NVIDIA Optimizing Compiler Library
Windows : nvvm.dll
MacOs : libnvvm.dylib
Linux : libnvvm.so

Component : NVIDIA Common Device Math Functions Library
Windows : libdevice.compute_20.bc, libdevice.compute_30.bc, libdevice.compute_35.bc
MacOs : libdevice.compute_20.bc, libdevice.compute_30.bc, libdevice.compute_35.bc
Linux : libdevice.compute_20.bc, libdevice.compute_30.bc, libdevice.compute_35.bc

Component : CUDA Occupancy Calculation Header Library
All : cuda_occupancy.h

Read more at: http://docs.nvidia.com/cuda/eula/index.html#ixzz30CrknWfU

Table of Contents

SIPro/PIPro 20

Contents 20

Getting Started with SIPro/PIPro 22

SIPro/PIPro Workflow 22

Analysis Capabilities 23

PI-DC Analysis 23

PI-AC Analysis 23

Power Plane Resonance Analysis 24

Power-Aware Signal Integrity Analysis 24

Design Assumptions 24

Example Workspace 25

Creating an SIPro and PIPro Setup 26

Creating a New SIPro/PIPro Setup 26

Opening an Existing SIPro/PIPro Setup 27

SIPro/PIPro Setup Window Elements 27

SIPro and PIPro Setup Window Overview 29

SIPro/PIPro Setup Window Overview 29

Contents 29

SIPro and PIPro Project Panel 29

Design 31

Nets 31

Changing Net Type 31

How to Find a Net 32

How SIPro/PIPro Identifies the Net Type 33

Setting the Net Type Manually 33

Components 33

How to Find a Component Instance 33
Substrate 34
How to View the Geometry by Layer 34
Pins 34

Definitions 34
Materials 34

Scripts 34

Analyses 35

Viewing and Hiding Project Tree Components 35
Customizing View 36

Workspace Windows 36

Layout Window 36

Simulation Window 39

Scripting Window 41
Scripting Options 41

Parameter Window 42
Parameter Options 43

Configuring SIPro and PIPro 43

General 44

Interface Tab 45

Modeling Tab 46

Nets 47

Add-on Manager 48

Add-on Manager 48
Add-on Search Path 49

Installing Additional Add-ons 49

Setup Options for Customizing Results 50

Range 50

Size Factor 52

Hiding and Unloading Results 53
See Also 53

Using Component Models 54

Using Component Models . . . 54
Contents . . . 54
Creating and Editing Component Models for Analysis . . . 54
Creating Component Models for Analysis . . . 54
Editing a Component Model for Analysis . . . 55
Adding a Lumped Model . . . 55
Adding an SnP Model . . . 56
Selecting an Existing Model DB . . . 57
Selecting Multiple Model DBs . . . 58
Specifying a Default Model . . . 59
Specifying a Model with more than 2 pins . . . 60
Specifying a Arrayed Component . . . 61
Using Vendor Parts DB Browser . . . 62
Using Vendor Parts DB Browser . . . 62
Adding Components to the Vendor Parts DB . . . 63
- Adding Custom Component . . . 64
  - Adding a Lumped Model . . . 64
  - Adding an SnP Model . . . 64

PIPro Analysis . . . 66

PIPro Analysis . . . 66
Contents . . . 66
Tutorial-Performing a PI-DC and PI-AC Analysis . . . 66
Performing a PI-DC Analysis . . . 67
- Creating a PI-DC Analysis Setup . . . 67
  - Defining a VRM . . . 67
  - Defining a Sink . . . 68
  - Defining Options . . . 70
  - Saving the Analysis Setup . . . 71
- Running the Analysis . . . 71
- Viewing Results . . . 71
  - Overview . . . 72
  - Voltage . . . 72
  - Current Density . . . 72
  - Power Loss Density . . . 73
  - Generate Schematic . . . 73

Performing a PI-AC Analysis 73
Enable MuRata Library 73
Opening the Example Design 74
Creating a PI-AC Analysis Setup 74
Defining a Component Model 75
Defining Options 76
Running the Analysis 77
Viewing PDN Impedance 77

PIPro Analysis Setup Overview 77

Prerequisites 77

Setup 77

VRMs 78

Sinks 78

Options 78

PI-DC Static IR Drop Analysis 79

PI-DC Static IR Drop Analysis 79

Creating a PI-DC Analysis Set up 79
Defining a VRM 80
Defining a Sink 80
Defining Options 81
Running the PI-DC Analysis 81
Show Me How Do I Perform a PI-DC Analysis 81

Viewing PI-DC Analysis Results 81
Running a PI-DC Analysis 81
Overview 82
Pins 82
VRMs 83
Vias 83
Reports 83
Setting Template 83
Voltage 83
Current Density 84
Power Loss Density 85
Generate Schematic 86

PI-AC Dynamic IR Drop Analysis 87

PI-AC Dynamic IR Drop Analysis 87

Creating a PI-AC Analysis Setup 87
Defining a PI-AC Analysis Setup 87
Copying the PI-DC Analysis Setup 87
Creating a New PI-AC Analysis Setup 87

Defining VRMs . . . 88
Defining Sinks . . . 89
Defining Component Models . . . 89
Defining Options . . . 90
Defining Frequency Plans . . . 90
Saving fields for field plots . . . 90
Saving the Setup . . . 91
Show Me How Do I Perform a PI-AC Analysis . . . 91

Viewing PI-AC Analysis Results . . . 91
Running a PI-AC Analysis . . . 91
PDN Impedance . . . 92
Viewing PDN Impedance . . . 92
Removing PDN Impedance Plots . . . 93
Importing a Target Impedance File . . . 93
Result Type Options . . . 94
VRM Options . . . 94
Viewing Electric Field . . . 95
Viewing Magnetic Field . . . 95
Current Density . . . 96
Generating a Schematic . . . 97
Enable the decoupling capacitors . . . 98
Tuning Parameters . . . 98

PI-PPR Analysis . . . 99

PI-PPR Analysis . . . 99

Creating a PI-PPR Analysis Setup . . . 99
Defining a PI-PPR Analysis Setup . . . 99
Copying the PI DC/AC Analysis Setup . . . 99
Creating a New PI-PPR Analysis Setup . . . 99
Defining VRMs . . . 99
Defining Sinks . . . 100
Defining Component Models . . . 100
Defining Options . . . 100
Defining Frequency Plan . . . 101
Saving the Setup . . . 101

Viewing PI-PPR Analysis Results . . . 101
Running a PI-PPR Analysis . . . 101
Electric Field . . . 101
Magnetic Field . . . 102
Current Density . . . 103

SIPro Analysis . . . 105

SIPro Analysis . . . 105

Contents . . . 105

Tutorial-Performing Power Aware SI Analysis . . . 105

Opening the Example Design . . . 105

Creating a Power Aware SI Setup . . . 106

- Defining Ports . . . 106
- Defining Options . . . 107

Running the Analysis . . . 108

Viewing Results . . . 108

- View TDR/TDT Results . . . 108
- Show Me How Do I Perform a Power Aware SI Analysis . . . 109

Creating a Power Aware SI Analysis Setup . . . 109

Creating a New Power Aware SI Analysis Setup . . . 109

Selecting Nets . . . 109

Defining I/O Ports . . . 110

Defining Component Models . . . 111

Defining Options . . . 111

- Defining Frequency Plans . . . 112

Saving the Setup . . . 112

Viewing Power Aware SI Analysis Results . . . 113

Running a Power Aware SI Analysis . . . 113

S-Parameters . . . 113

Using the S-Parameters Results Window . . . 114

- Data . . . 114
- Insert . . . 114
- View . . . 115
- Toolbar Options . . . 115
- Graph Type Selector . . . 116
- Single Ended Tab . . . 116
  - Port Property Table . . . 116
- Mixed Mode Tab . . . 119
  - Port Property Table . . . 119
  - Context menus . . . 120
    - Context menus only for Mixed Mode tab . . . 120
    - Context menus not available for Mixed Mode tab . . . 120
    - View Mixed Mode S-parameters . . . 120

View TDR/TDT Results . . . 121

Using the TDR/TDT Results Window . . . 121

- Export . . . 121
- Toolbar Options . . . 122

TDR/TDT Setup Options 122
Single Ended Tab 122
Port Property Table 122
Mixed Mode Tab 123
Port Property Table 123
Matrix Selector 123

Generate Sub Circuit 124

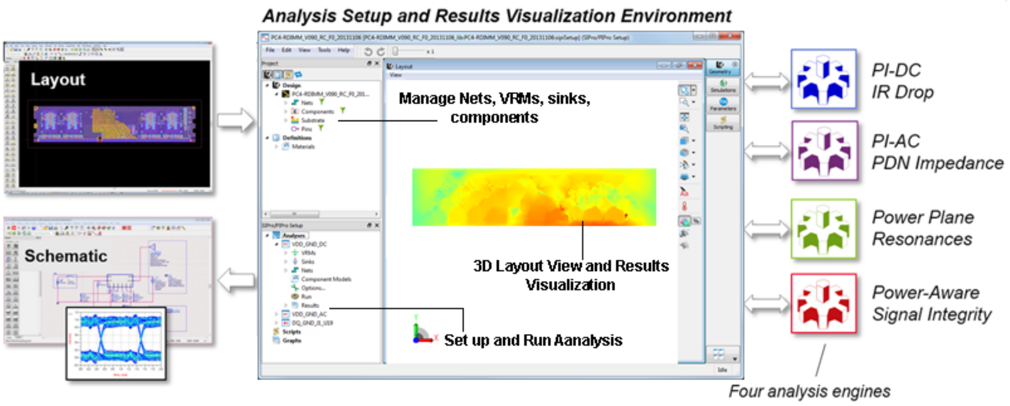
Installing the Deprecated SI and PI Analysis Addon 125

Installing the Deprecated SI and PI Analysis Addon 125

SIPro/PIPro Videos 127

SIPro/PIPro

SIPro/PIPro is a simulation and analysis tool that enables you to evaluate the signal integrity (SI) performance of signal nets and the power integrity (PI) performance of power distribution networks (PDNs). This tool provides several capabilities to perform pre-layout analyses and post-layout verifications. The following figure illustrates the analysis setup and results visualization environment of SIPro/PIPro:



Contents

- Getting Started with SIPro and PIPro
- Creating an SIPro and PIPro Setup
- SIPro and PIPro Setup Window Overview
- Using Component Models
- PIPro Analysis
    - PIPro Analysis Setup Overview
    - Tutorial-Performing a PI-DC and PI-AC Analysis
    - PI-DC Static IR Drop Analysis
        - Creating a PI-DC Analysis Setup
        - Viewing PI-DC Analysis Results
    - PI-AC Dynamic IR Drop Analysis
        - Creating a PI-AC Analysis Setup
        - Viewing PI-AC Analysis Results
    - PI-PPR Analysis
        - Creating a PI-PPR Analysis Setup
        - Viewing PI-PPR Analysis Results

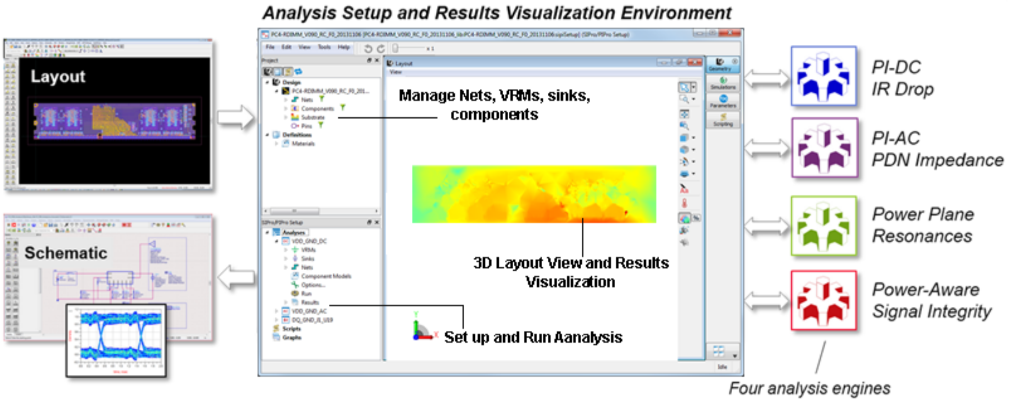
- SIPro Analysis
  - Tutorial-Performing Power Aware SI Analysis
  - Creating a Power Aware SI Analysis Setup
  - Viewing Power Aware SI Analysis Results
- SIPro and PIPro Videos

Getting Started with SIPro/PIPro

Using the SIPro/PIPro simulation and analysis tool, you can evaluate the signal integrity (SI) performance of signal nets and the power integrity (PI) performance of power distribution networks (PDNs). It enables you to perform pre-layout analysis and post-layout verifications. SIPro/PIPro provides the following analysis capabilities:

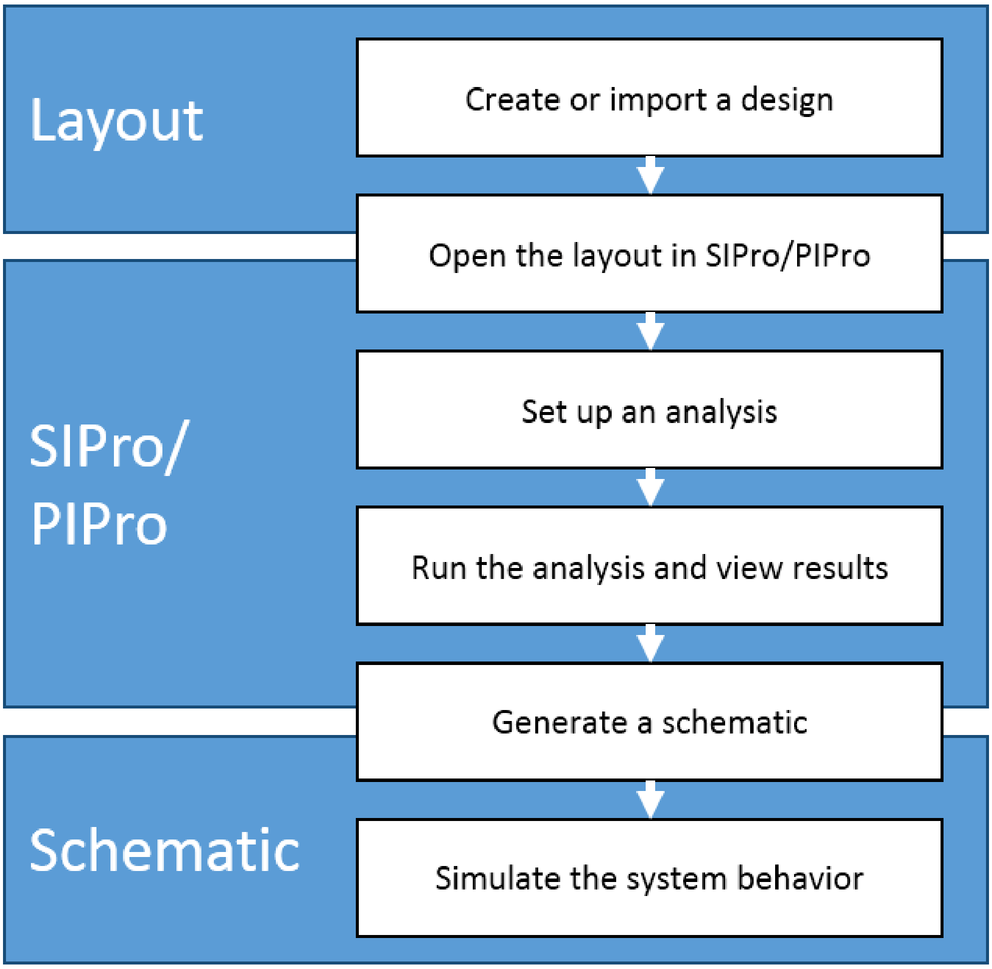
- PI-DC Analysis
- PI-AC Analysis
- Power Plane Resonance Analysis
- Power-Aware Signal Integrity Analysis

The following figure illustrates the analysis setup and results visualization environment of SIPro/PIPro:



SIPro/PIPro Workflow

The workflow for evaluating the SI or PI performance of a layout design is displayed in the following figure:



Analysis Capabilities

PI-DC Analysis

A PI-DC analysis computes the voltage, IR drop (voltage drop), current, and power loss density in the power supply nets. It helps you to identify the IC and connector pins or stitching vias drawing large amounts of current at DC operating conditions. Due to excessive voltage drop, the power supply voltage at the IC might fall below the recommended minimum voltage. This can cause malfunctioning of the IC. Excessive current density in the perforated power supply rails can generate excessive heat, which might lead to board failures due to delamination or fusing. Also, excessive current in the stitching vias can lead to via failures losing connection. Any number of power supply nets with source and sink models can be simulated together.

PI-AC Analysis

A PI-AC analysis computes the impedance for the IC current loads over a broad frequency range. It helps you to identify whether the power distribution network (PDN) provides a low impedance path from the Voltage Regulator Module (VRM) to the IC. An excessive impedance in a certain frequency range can generate excessive voltage noise, also called dynamic IR drop, when the IC power supply pins draw large amounts of transient current, required for I/O or core logic switching, at rates that fall into that frequency range.

Power Plane Resonance Analysis

A power plane resonance (PPR) analysis computes the self-resonant frequencies and corresponding Q-factors of the power distribution network (PDN). It helps you to identify optimal placement of ICs, decoupling capacitors and stitching vias. A power plane resonance can disturb sensitive analog circuitry and generate excessive radiation. This can cause that EMC specifications cannot be met.

Power-Aware Signal Integrity Analysis

A power-aware signal integrity analysis computes a model characterizing the behavior of signal and power networks. The model can be assesed from within the SIPro window and can be used as input for further analysis in circuit simulation, e. g. channel or transient simulations.

NOTE For PI-AC, Power Plane Resonance Analysis and Power-Aware Signal Integrity Analysis, the minimal recommended memory requirement is 4 GB, preferrably higher. There is a fixed overhead cost even for small designs of 1.5-2GB. The memory growth as the simulated designs get larger is close to linear. The memory requirement is not dependend on the requested frequency range.

Design Assumptions

The recommended starting point for using SIPro and PIPro is **a layout with instantiated components**. A flat layout with top level pins can be used, but an analysis setup is much easier when the component instances are available. **Net names** play a key role in the analysis setup. The file import in ADS for following design transfer formats preserves the net names when that information is provided by the third party tool. Verify the file export options in the third part tool to pass as much design information as possible.

Vendor	**Tool**	**Recommended Design Transfer Format**
Altium®		
	Designer	ODB++
Cadence®		
	Allegro PCB	BRD or ODB++
	APD	ADFI
	SiP	ADFI
	OrCAD	ODB++

Mentor Graphics®		
	Expedition	ODB++
	PADS	ODB++
	BoardStation	ODB++
Zuken™		
	CR5000	ODB++
	CR8000	ODB++
	CADSTAR	ODB++
Other Formats		
		ABL

The **substrate** (layer stackup) defines the arrangement and materials of the signal and power plane layers in a multi-layer board or package design. Always verify the substrate definition in case of a design transfer from a third party tool. The third party tool often does not export the full substrate specification. Once you have the layout with components, net names and substrate, you are ready to open the SIPro/PIPro Setup window.

Example Workspace

An example workspace to get started with SIPro and PIPro is provided with ADS, see **examples/HSD/SIPro_PIPro_Getting_Started_Example_wrk.7zads**. The workspace contains a Samsung DDR3 UDIMM memory card. The design files are from the JEDEC (www.jedec.com). The design consists of a 6 layer board with single power rail for core and I/O buffers.

Creating an SIPro and PIPro Setup

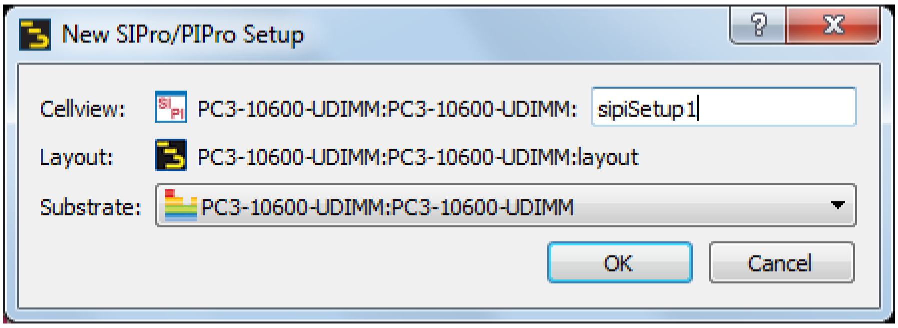
To analyze the SI and PI performance, you can create an SIPro/PIPro setup in the following ways:

- Create a new setup
- Open an existing setup

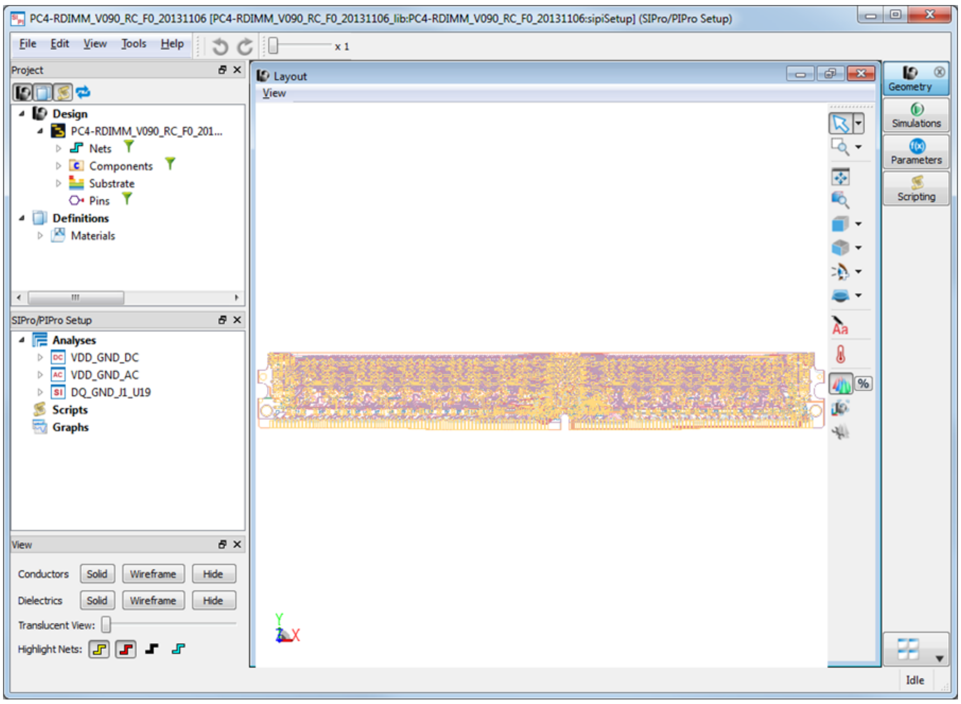
Creating a New SIPro/PIPro Setup

To create a new SIPro/PIPro setup:

1. Open a Layout window in ADS.
2. Select **Tools** > **SIPro/PIPro** > **New Setup** from a Layout window to create a new setup. The New SIPro/PIPro Setup window is displayed, as shown in the following figure:
3. Specify the **Cellview** name.



4. Select the required substrate.
5. click **OK**. A new SIPro/PIPro Setup window is displayed, where you can set up and run an analysis.



NOTE

The SIPro/PIPro window cannot be used for ADS layout designs that contain the following features:

- Derived layers
- 3D EMPro components
- Slot layers
- Multi-technology setup

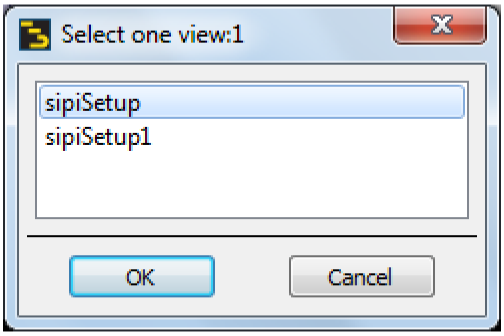
In addition, the SIPro/PIPro setup assumes meaningful and consistent net definitions in the design.

Opening an Existing SIPro/PIPro Setup

The SI and/or PI analysis setup data for a specific design is stored in a cell view of the “SIPro/PIPro Setup” type . The default view name is “sipiSetup”. These views are registered with OA and behave like “Schematic” and “Layout” views. You can perform various tasks such as, renaming, copying, moving, and archiving. A single view can contain multiple analysis setups, such as PI-DC and PI-AC analysis setup.

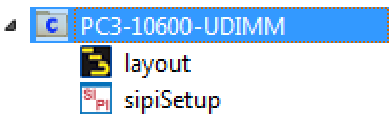
To open an existing setup:

1. Open a Layout window in ADS.
2. Select **Tools > SIPro/PIPro > Open Setup** from a Layout window to open an existing setup. The Select one view window is displayed, as shown in the following figure:
3. Select the required view.



4. Click **OK**. The SIPro/PIPro Setup window is displayed.

Alternatively, you can open an existing setup by clicking the sipiSetup view in the Main window.



SIPro/PIPro Setup Window Elements

The SIPro/PIPro Setup window consists of the following elements:

GUI Element	Description
Project Panel	Consists of a panel that provides a tree-structured representation of the design Parts, Definitions, Graphs, and Analyses.
Geometry Window	Comprises the main viewing area. It enables you to perform various viewing operations.
Simulation Window	Allows you to monitor analyses sent the calculation engine.
Parameters Window	Enables you to create, edit, and delete parameters that can be referenced in an analysis setup
Scripting Window	Allows you to view, edit, and execute scripts.
Menus	Provides File, Edit, View, Tools, and Help menus.
Workspace Tabs	Provides Geometry, Simulation, Parameters, and Scripting tabs to control the associated windows

For more information about SIPro/PIPro windows, see Workspace Windows.

SIPro and PIPro Setup Window Overview

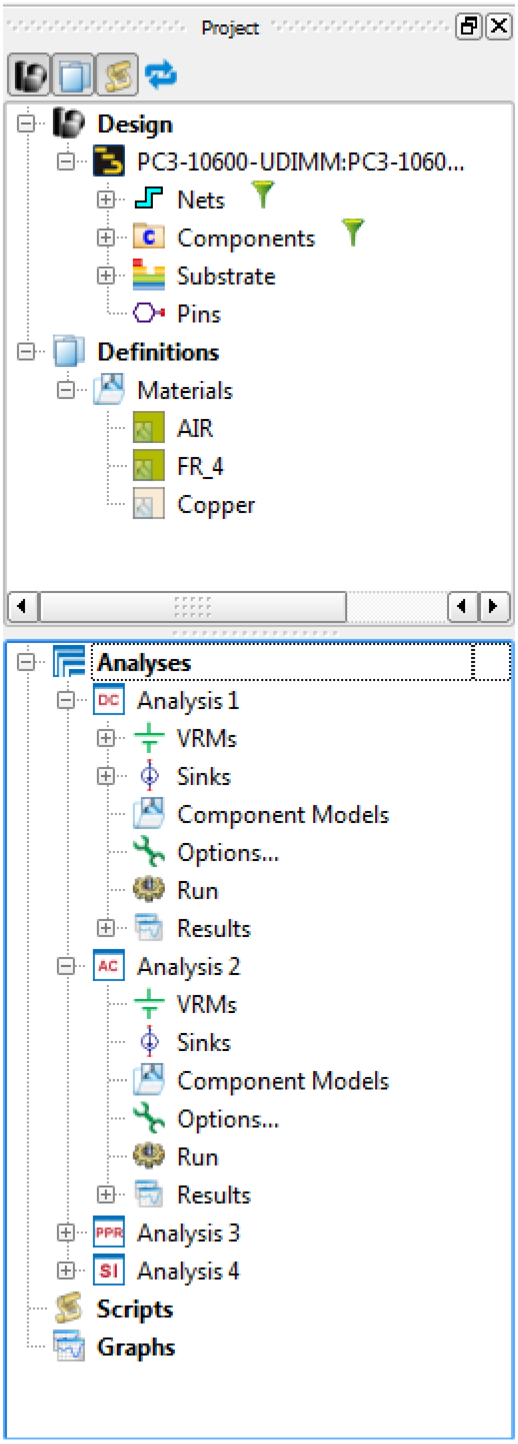
SIPro/PIPro Setup Window Overview

Contents

- SIPro and PIPro Project Panel
- Workspace Windows
- Configuring SIPro and PIPro
- Add-on Manager
- Setup Options for Customizing Results

SIPro and PIPro Project Panel

In the SIPro/PIPro Setup window, the Project panel provides a tree-structured representation of a design. You can use the Project panel to define a specific PI or SI analysis setup, to run the analysis and to review results. The Project panel toolbar enables you to show or hide specific items of a design. The following figure displays a project panel:

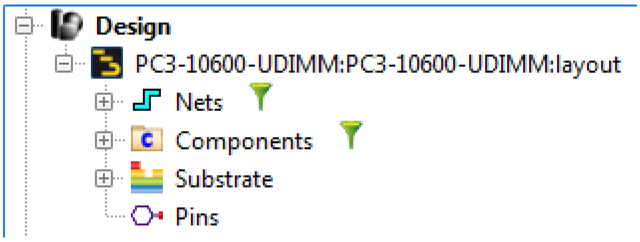


A project panel consists of the following components:

- Design
- Definitions
- Analyses
- Scripts
- Graphs

Design

The **Design** list displays the physical parts of the design-under-test. Currently, this is the layout view from which the SIPro /PIPro Setup was opened, no other parts can be added. The layout tree contains four items: **Nets**, **Components**, **Substrate** and **Pins**.



Nets

All nets in the layout view are listed. Each net gets a type, ‘Power’, ‘Ground’, ‘Signal’ or ‘Undefined’ assigned. The nets are sorted by type.

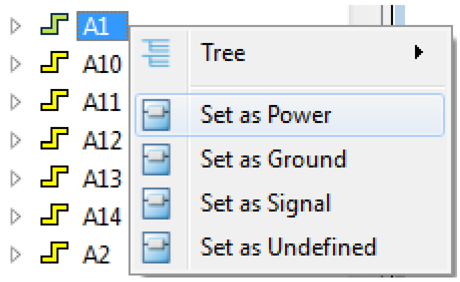
Type	Icon
Power	
Ground	
Signal	
Undefined	

NOTE Verify that the nets that you plan to simulate have been typed correctly. Even though SIPro /PIPro attempts to automatically identify the net type, the algorithm can miss nets with arbitrary names.

Changing Net Type

To change a net type:

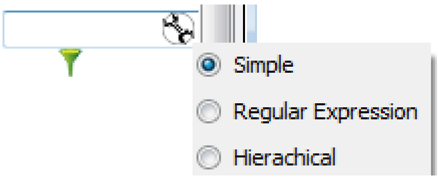
1. Select the required net.
2. Right-click the selected net and select the required type.



How to Find a Net

To find a specific net:

1. Click the **Filter** icon ( ) next to **Nets** in the **Parts** tree.
2. Type the required net name.
3. Click to display a list of matching algorithms, as shown in the following figure:



   - **Simple**: Display tree nodes that contains specified text
   - **Regular Expression**: Display tree nodes that contains specified regular expression.
   - **Hierarchical**: Display tree nodes that contains specified regular expression per hierarchy level separated by slash.

   Regular Expression is very powerful tool but it requires certain knowledge to use. Below is some convenient example for typical use cases.

Example	Match	Note
U(1|2|5|11)	Matches U1, U2, U5 and U11	"|" means "OR"
U[5-8]	Matches U5, U6, U7 and U8	"-" can be used for "range"
U1[1-3]	Matches U11, U12 and U13	U[11-13] doesn't work as expected
DQ[1-8]N?	Matches DQ1, DQ1N, DQ2, DQ2N,...,DQ8 and DQ8N	"?" can be used as optional string

4. Select the string matching algorithm: Simple, Regular Expression, or Hierarchical.

NOTE The Hierarchical option is relevant when tree nodes are hierarchical

How SIPro/PIPro Identifies the Net Type

SIPro/PIPro attempts to identify the net type by case-insensitive name matching. In case no match is found, the net type is 'Undefinded'. The algorithm uses following rules by default:

Type	Regular Expression	Examples
Power	pwr|power|vdd|vcc|vref|vtt|bat ^[\+\-]?\d+[p_]\d+v ^[\+\-]?\d+v ^[\+\-]?\d+	AVDD, DVDD, VREF, VTT, VBAT +1p2v, -1p2v, 1p2v, +1_2v, -1_2v, 1_2v +12v -12v 12v +12, -12, 12
Ground	gnd|ground|grnd|vss	AGND, AVSS
Signal Memory Clock Diff pair HS Serial	dq a\d+ ba\d+ clock|clk _[pn]$ [\+\-]$ sig|tx|rx	DQ0, DQ1 A01, A02 BA01, BA02 CLK DQS0_P, DQS0_N DQS0+, DQS0- SIG01

This rule can be modified by user. See here for more detail.

Setting the Net Type Manually

To override the automatic net type, you can select one or more nets, right-click and choose the appropriate net type.

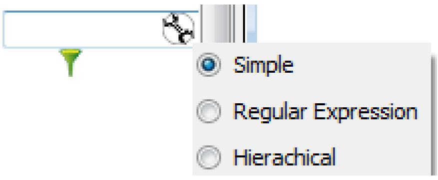
Components

All component instances in the layout view are listed and grouped per component.

How to Find a Component Instance

To find a specific component instance:

1. Click the **Filter** icon () next to **Components** in the **Parts** tree.
2. Type the component instance name.
3. Click  to display a list of matching algorithms, as shown in the following figure:



4. Select the string matching algorithm: Simple, Regular Expression, or Hierarchical.

Substrate

The Substrate tree item presents the objects in the layer grouped by Conductor layer, Via layer or Dielectric layer.

How to View the Geometry by Layer

First, switch off the Parts Visibility toggle. This toggle can be found on the Geometry Window's View toolbar. Then, select the appropriate layer in the Substrate tree. The view displays the objects on the selected layer only.

Pins

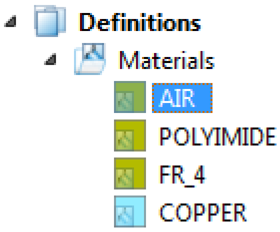
All top level pins in the layout are listed.

Definitions

The *Definitions* branch stores definitions that can be applied to or shared with other objects within the project. You can apply a definition to other objects in the Project panel by clicking and dragging the definition object on the required object.

Materials

To view a material definition object, select a material from the **Materials** list.

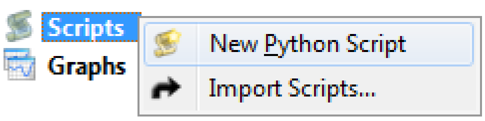


Scripts

The *Scripts* branch stores user-defined scripts. Right-click **Scripts** to add a new script or import an existing macro or function script to the project.
You can execute or edit the script in the Scripting workspace window.

To add a new scripts:

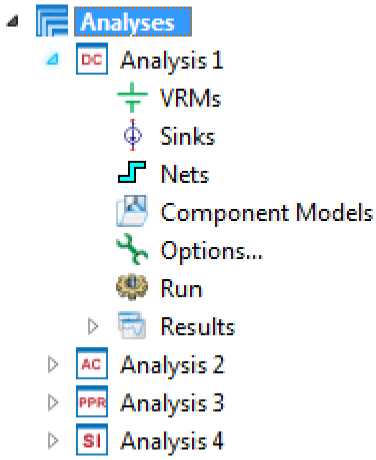
1. Right-click **Scripts** in the Project panel:



2. Select **New Python Script** to create a new script or click **Import Scripts** to use an existing script.

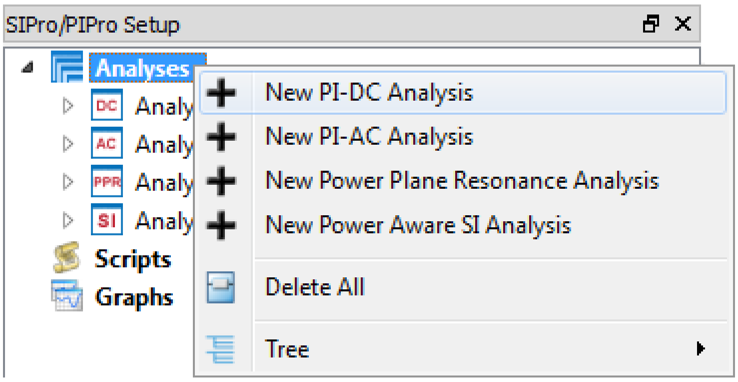
Analyses

The Analyses branch enables you to run a PI DC analysis, PI AC analysis, PPR analysis, and SI analysis. By default, it provides an empty template for creating an analysis, as shown in the following figure:



Instead of using the default templates, if you want to add a new analysis:

1. Right-click **Analyses** in the Project panel:



2. Select the required analysis. A new analysis is added at the end of the Analysis branch.

Viewing and Hiding Project Tree Components

You can view or hide options present in the Project panel by using toggle buttons. The following figure displays these toggle buttons:

The following table describes each toggle button:

Option	Icon	Description
View Parts		Enables you to view or hide **Parts** in the project panel.
View Definitions		Enables you to view or hide **Definitions** in the project panel.

Option	Icon	Description
View Scripts		Enables you to view or hide **Scripts** in the project panel.
Toggle All		Enables you to view or hide all the components of the project panel.

Customizing View

To set a specific view for your design in the layout window, you can right-click the required component and select one of the following options:

Option	Icon	Description
Set Visible		Enables you to view the selected component in the Layout window.
Set Invisible		Enables you to hide the selected component in the Layout window.
Solid		Enables you to display components in a solid view in the Layout window.
Wireframe		Enables you to view or hide all the components of the project panel.
Delete		Enables you to delete a component.
Expand		Enables you to display all the subparts of a component in the project panel.

Workspace Windows

The SIPro/PIPro Setup window consists of the following windows:

- Layout
- Simulation
- Parameters
- Scripting

Layout Window

The Layout window provides a layout view of your design. In this window, you can use various tools for customizing the view of a simulation space. This window comprises the main design viewing area.

The following table describes options available in the View menu and View Tools toolbar:

Option	Icon	Description
Toolbars		• View Tools
View Manipulation		• **Select**: The Select tool is used to select objects as well as manipulate the view of the simulation space. • **Orbit**: The Orbit tool is selected to perform rotation of the simulation space through left-clicking-and-dragging. • **Pan**: The Pan tool is selected to perform translation of the simulation space through right-clicking-and-dragging. • **Zoom**: Zoom-in or zoom-out of simulation space by scrolling up or down the mouse wheel, respectively. • **Zoom to Window**: Zoom into a rectangular shaped area of the geometry as specified by the user. To use, select the tool, then left-click and drag the mouse to designate the rectangular zoom area.
Zoom to Extents		Enable you to zoom automatically as can view the entire geometry.
Zoom to Selection		Enables you to automatically zoom selected object only.
Standard Views		The Standard View button function is to automatically change the perspective of the objects in the workspace window.

Option	Icon	Description
		• Front (-Y) • Back (+Y) • Top (-Z) • Bottom (+Z) • Right (-X) • Left (+X)
Isometric Views		The Isometric Views button function is to automatically change the perspective of the objects in the workspace window. • Front/Right/Top • Front/Left/Top • Front/Right/Bottom • Front/Left/Bottom • Back/Right/Top • Back/Left/Top • Back/Right/Bottom • Back/Left/Bottom
Custom Views		Enable you to define any perspective than the Standard Views and Isometric Views, and display at that perspective. • Add View
Cutting Planes		Cutting Planes button feature allows you to define an arbitrary cross-section of the object and display it. • Toggle Cutting Plane • Edit Cutting Plane • Save Cutting Plane
Field Reader		The Field Reader tool measures field values at the location where the mouse hovers over the geometry.

Option	Icon	Description
Export Image		The Export Image tool takes a screen shot of the geometry as it is currently shown in the workspace window, and saves it to a specified location.
Toggle Parts Visibility		Toggles the design parts on and off. Clicking the *Opacity* button located to the right of this button, opens a slider to customize the translucency of its objects. The sliders change the alpha of the design objects, making them more or less translucent as the slider is dragged right or left, respectively.
Toggle Output Visibility		Toggles the output results on and off. Clicking the *Opacity* button located to the right of this button, opens a slider to customize the translucency of its objects. The sliders change the alpha of the design objects, making them more or less translucent as the slider is dragged right or left, respectively.
Toggle Bounding Box Visibility		Ttoggles the visibility of the bounding box of a design when the design is selected.
Toggle Output Viewing Controls		Toggles the visibility of the output viewing controls for sensor results.

Simulation Window

The Simulations workspace provides an interface to define simulations to send to the calculation engine. Simulations can be easily created, defined, and stored in the Simulations workspace window. Each time a project is modified and saved, you must define a new simulation to register the change. This workspace window stores definitions such as source types, parameter sweeps, S-Parameters, frequencies of interest, scattered/total field interfaces and termination criteria that are specific to a calculation. EMPro supports the following solver EM simulation technologies:

The following table lists the icons present in the Simulations window:

Option	Icon	Description
Simulation		Starts a simulation.
Add To Queue		Selects the required simulation from the upper panel of the Simulations window and add to the queue.
Remove From Queue		Selects the required simulation from the upper panel of the Simulations window and remove from the queue.
Delete From Queue		Deletes the selected simulation from the disk.
To start the next phase		To interrupt the current phase of the simulation and start the next phase.
Kill the Queue		Kills the currently executed simulation.

The following information is available in the upper pane of the Simulations window:

Option	Description
Id	Displays a project ID, which references the location of the loaded project in the file directory.
Name	Displays the project name specified by a user.
Date Created	Specifies the date when a simulation was created.
Engine	The simulator used for performing simulation.
Status	Displays whether a result is complete or still being calculated while the simulation is running. This status can be refreshed manually by pressing the Update button or automatically by selecting Auto-scroll.

The following tabs are available in the lower pane of the Simulations window:

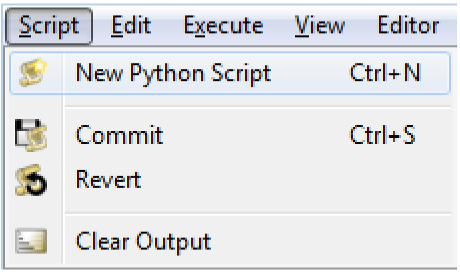
Option	Description
Summary	Provides a summary of the simulation such as simulation path, status, boundary conditions, ports, feed, and waveforms.
Notes	Displays any notes that you might have added while setting up a simulation.
Log	Displays a log of tasks performed during the simulation.

Scripting Window

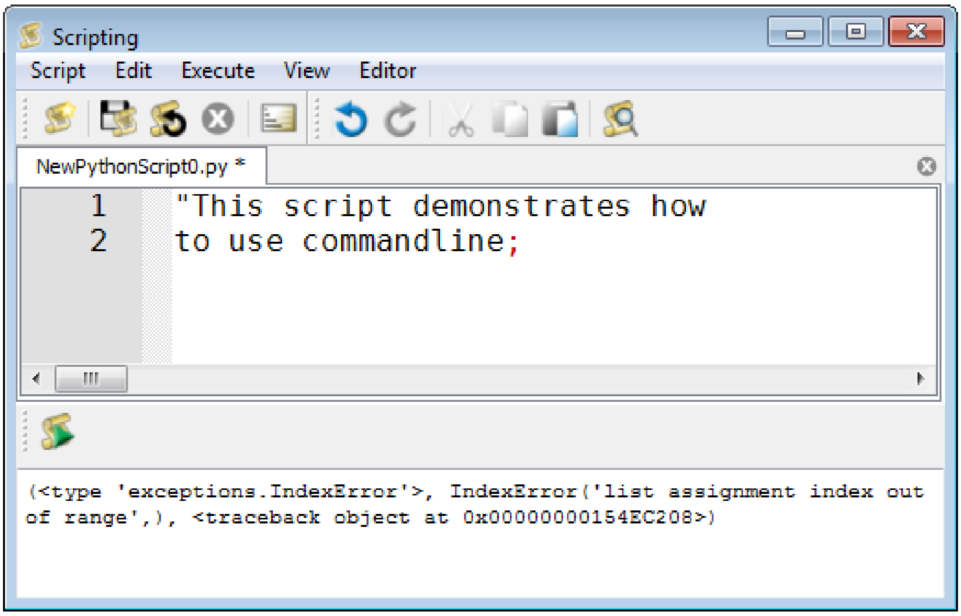
The Scripting workspace allows you to view, edit, and execute scripts.

To create a new python script;

1. Select **Script** > **New Python Script**, as shown in the following figures:



2. Type commands in the scripting window:



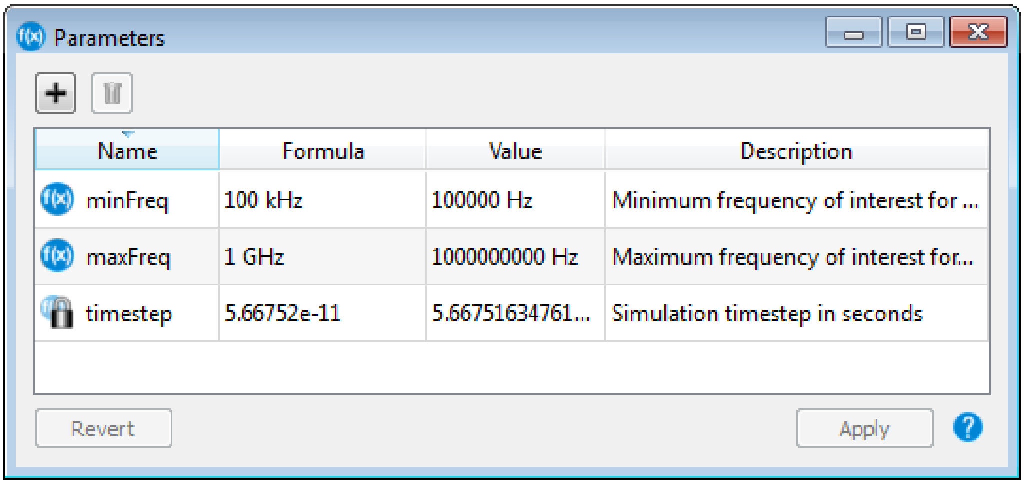
Scripting Options

The following table describes options available in the Scripting window toolbar:

Option	Description
Script	• New Python Script • Commit • Revert • Clear Output
Edit	Enables you to copy, cut, or select text in a script.
Execute	• Execute Script • Cancel Execution
View	• Script Tools • Editing Tools • Execution Tools
Editor	• View • Indentation • ConvertEOL • PsychicPy

Parameter Window

The *Parameters* window enables you to create, edit, and delete parameters that are referenced. By default, three parameters are provided **minFreq**, **maxFreq**, and **timestep**.



To modify the frequency range of the default minimum and maximum frequency parameters:

1. Click **Parameters**. The Parameters window is displayed.
2. Double-click the **minFreq** or **maxFreq** value in the **Formula** field.
3. Specify the required value.
4. Click **Apply**.

Parameter Options

The following table describes the Parameters options:

Name	Description
Name	Enables you to define a parameter, which can be referenced later.
Formula	Allows you to provide a mathematical formula or a simple numeric value that will define the value of a given parameter. This formula can reference other parameters that have already been defined.
Value	This column is a read-only column that displays the evaluated value of the parameter. If an invalid formula is entered, an error message will appear within this field with a description of the invalidity. Simply hold the mouse over the error message to view this description. Similarly, a parameter is deleted by selecting the unwanted parameter and clicking the button above the table. If a parameter is deleted that is referenced within another parameter's definition, an error message will appear since the parameter that is referenced is no longer defined.
Description	Allows you to describe a parameter.

Configuring SIPro and PIPro

Before setting up an SIPro/PIPro project, you can configure the project settings by using the Options window. This window consists of various tabs such as General, Interface, Modeling, and Nets. Using these tabs, you can specify the global settings for your project.

Select **Tools**> **Options** to open the Options window. In this window, you can use the following tabs:

- General
- Interface
- Modeling

- Nets

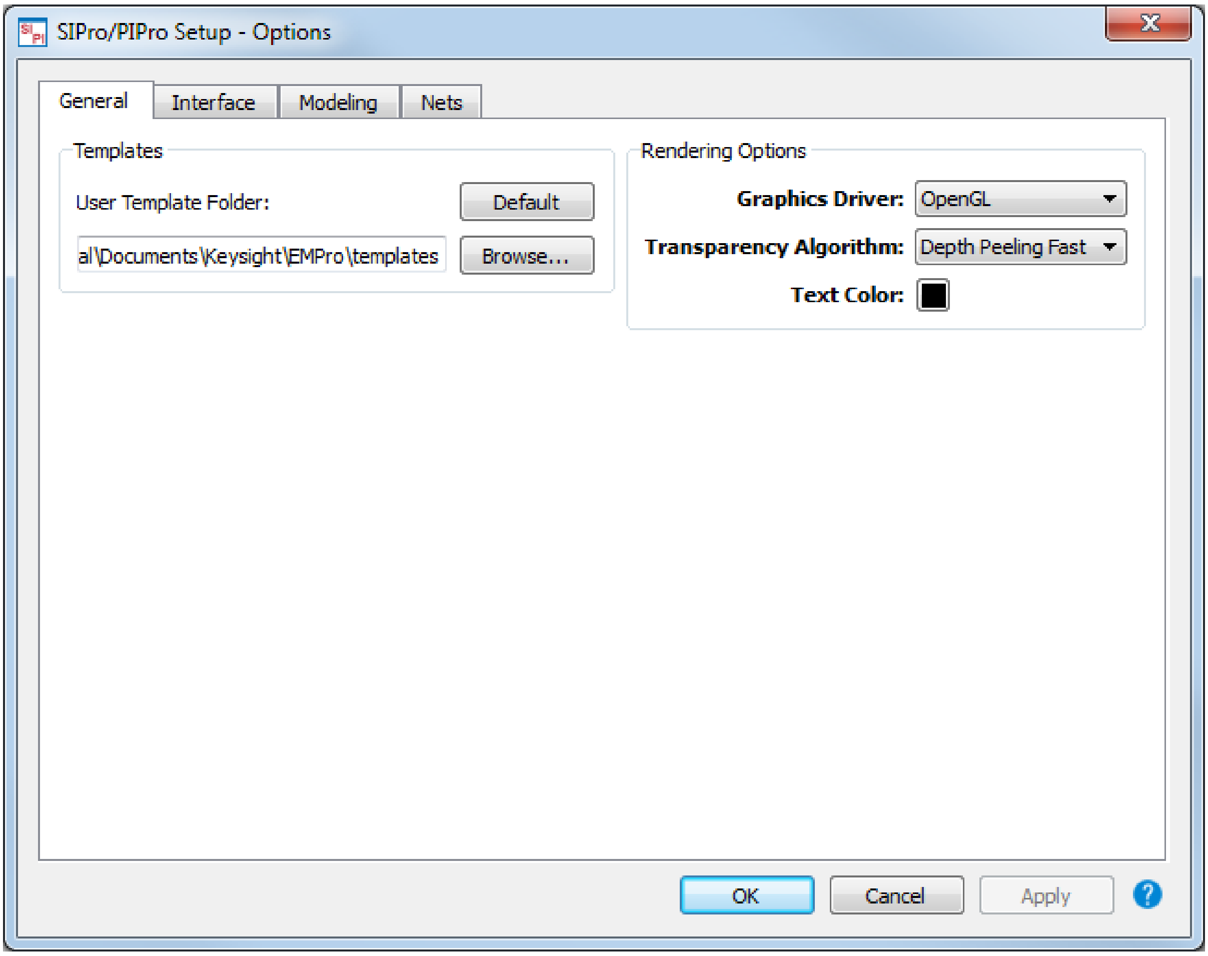
General

The General tab enables you to set the following options:

- **Templates**: Enables you to specify the default folder location for storing SIPro/PIPro project templates.
- **Rendering Options**: Allows you to control several display options in the SIPro/PIPro GUI:
- **Graphics Drivers**: Specifies the display driver to be used for rendering graphics in SIPro/PIPro. The list of available drivers depends on the operating system, graphics card, and on the usage of remote desktop. Using an unsupported display driver may result in a blank Geometry Window. Any change to this setting will take effect until you restart SIPro/PIPro. The display driver can also be changed using the --driver command line option when starting SIPro/PIPro. Available graphics drivers are: OpenGL, OpenGL 2, DirectX 9, X11, MSW.
- **Transparency Algorithm**: Determines the way SIPro/PIPro renders the parts opacity of an object. You can select the values:
    - **Z-sort Only Fast**: This is the most efficient rendering option, but may contain artifacts
      (small areas of inaccurate rendering).
    - **Z-sort Only Nice**: This is a more accurate rendering option than Z-sort Only Fast, but runs slower. It will have less artifacts.
    - **Depth Peeling**: This option is more accurate than both Z-Sort algorithms. It's performance is dependent on the graphics card, and may be significantly slower if the card does not support it.
    - **Painters Algorithm**: This is the most accurate rendering option, but runs the slowest. On modern graphics cards, this option works very well.

    NOTE

    Transparency Algorithm: This setting affects only the way an object is displayed in the SIPro/PIPro interface, and does not affect calculation results.

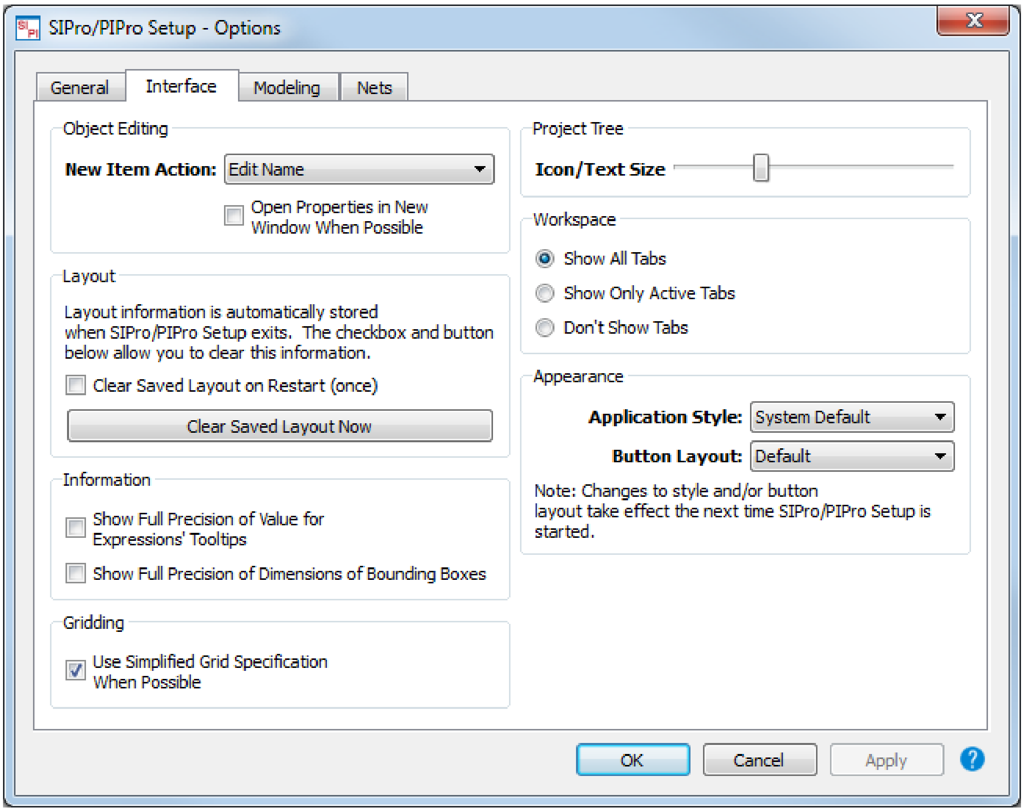


Interface Tab

The Interface tab enables you to set the following options:

- **Object Editing**: Controls the editing of objects, when a new object is added to the project tree by right-clicking the project tree. You can select one of the following values from the New Item Action drop-down list:
  - None: Adds a new object to the project tree.
  - Edit Name: Adds an object to the tree and provides a blinking cursor to add the object name in the project tree.
  - Edit Properties: Adds an object and displays the appropriate window to edit the new object properties. The Material Editor dialog box is displayed to specify settings.
- **Layout**: Enables you to set preferences for saving or restoring the GUI layout.
- **Information**: Enables you to to set their preference for decimal precision in tooltips.
- **Gridding**: Enables or disables the use of the simplified grid.
- **Project Tree**: The *Icon/Text Size* scroll adjusts the size of the items in the project tree.
- **Workspace**:
  - Show All Tabs shows all workspace windows in tabbed workspace regardless of whether they are active or not.
  - Show Only Active Tabs stores only the active tabs that are stored in the project workspace.
  - Don't Show Tabs removes the tabbed workspace. Windows can still be accessed from the View menu.

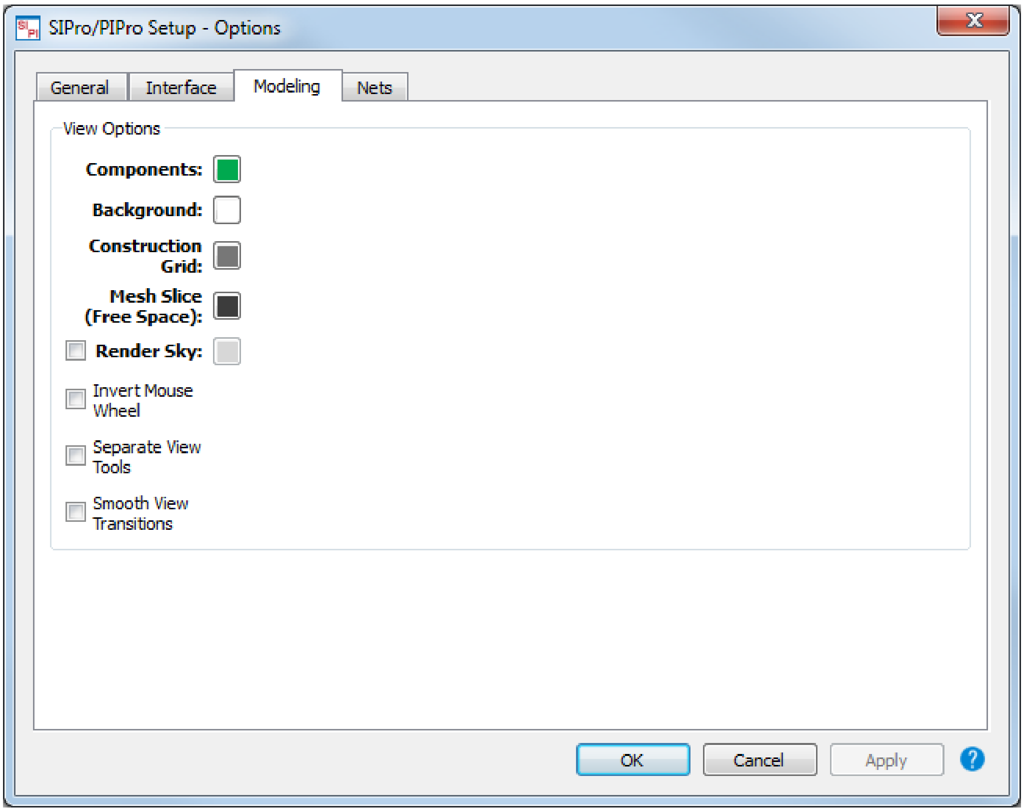
- **Appearance**: The Appearance pane allows you to change the appearance of the SIPro/PIPro GUI and buttons within.



Modeling Tab

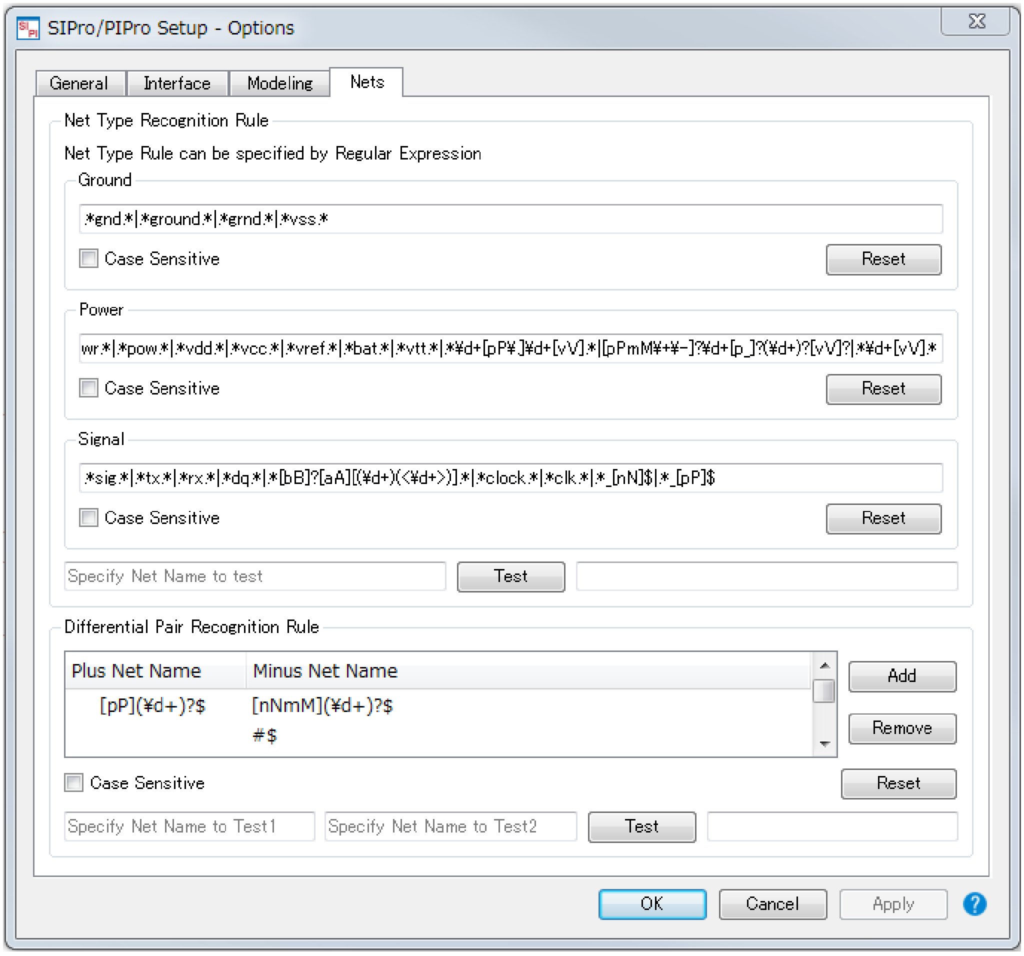
The Modeling tab provides several options for adjusting the color and appearance of faces, edges, vertices, components, and the background. You can set the default colors used to display faces, edges, and/or vertices for parts when they are first created. You can also set the color of the components (like voltage sources) and Geometry window background color.

- Components: Specifies the color used to render circuit components.
- Background: Specifies the background color of the View.
- Construction Grid: Specifies the color for the lines of the construction grid of the Sketcher and Orientation tools.
- Mesh Slice: The color for mesh edges that are Free Space.
- Invert Mouse Wheel: Switches the zoom direction when rolling the mouse wheel.
- Separate View Tools: Toggles between showing all View tools in a drop-down list or as individual items.
- Smooth View Transitions: Specifies SIPro/PIPro to perform smooth rotations between different view orientations. If you do not select this option, SIPro/PIPro snaps to the selected view orientation.



Nets

Using the Nets tab, you can specify the net type recognition rules for ground, power, and signal by using Regular Expression options.



Also, you can specify differential pair recognition by using pairs of Regular Expression for plus net and minus net. Differential Pair recognition is currently used only for mixed mode of S-parameter plots and TDT/TDR plot window.

Specify regular expression for Plus Net and Minus Net respectively. If given any of two Net names are matched with given regular expressions and rest string that are matched pieces are stripped off are the same, they are recognized as differential pair.

For example, regular expression P$ for Plus and N$ detect Sig1P and Sig1N as differential pair.

Either one of Plus or Minus can be empty. This can be used to recognize a case such as pair name is like AAA and AAA#. (empty regular expression for Plus Net and #$ for Minus Net).

Add-on Manager

You can extend the SIPro/PIPro user interface with additional functions, which are customized for your design needs. These tools are python scripts that you can customize for your workflow. You can access the following add-on tools:

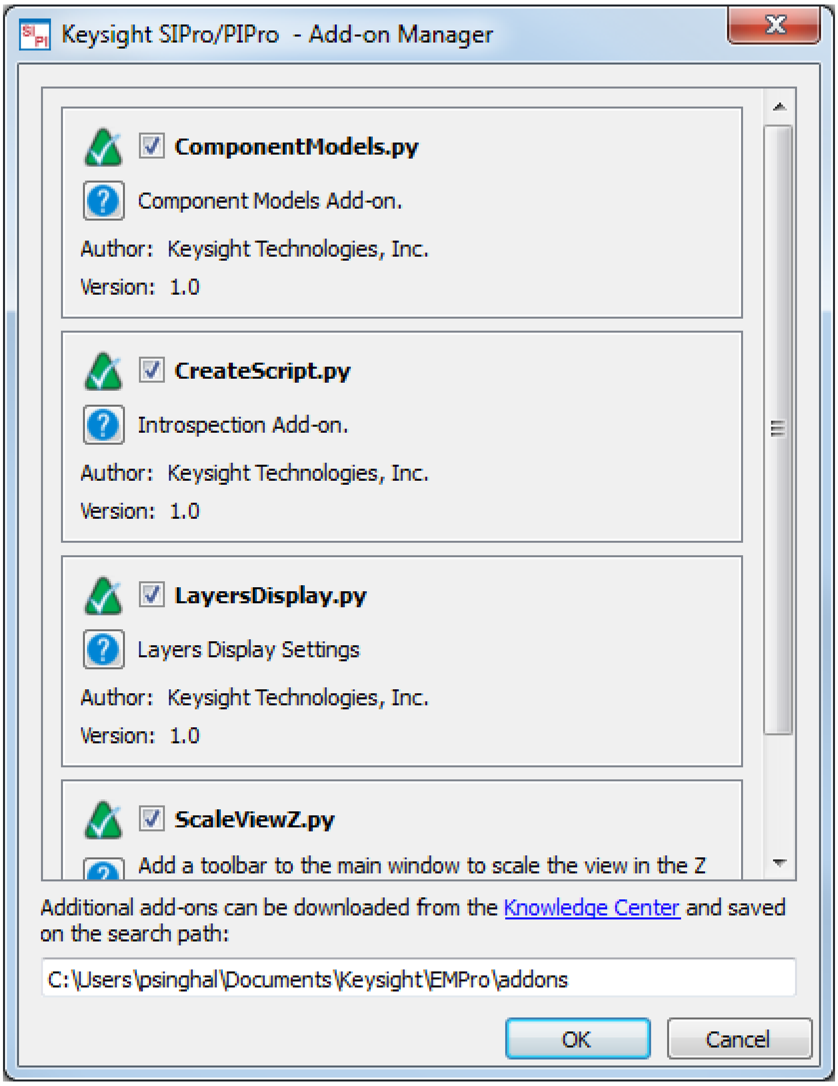
- Component Models Add-on: Enables you to Export their Components and ComponetModels to a file and also allows to Import them.
- Introspection Add-on: Enables you to generate the recipie of the setup and the elements of the setup like Sinks, Vrms, Component Models and Results.
- Layers Display Setting: Enables you to change the Display Settings of the Conductive layers and the via layers in the Active Project.
- ScaleViewZ Add-on: Enables you to add a toolbar to the main window to scale the view in the Z direction.

Add-on Manager

The Add-on Manager allows you to enable or disable add-ons, inspect their status and possible load errors, and modify the search path. It can be accessed from **Tools** > **Add-on Manager**.

The Add-on Manager displays a separate panel with the following elements:

- A check box with the file name of the add-on: Select the check box to enable a add-on. Hovering over the file name will show the full path of the add-on.
- The load status on the left of the check box: A green icon is shown when an add-on is successfully loaded, a yellow exclamation mark when it failed to load. Hover over the icon to see the full error message. No icon will be shown for a disabled add-on that is not loaded.
- If available, a short description of the add-on: Click  in the Add-on Manager to display more information about an add-on.
- If available, two labels with the author and version information of the add-on.



Add-on Search Path

You can download additional add-ons, which are saved on the following search path:

- **Additional search path**: A user specified location where downloaded or custom add-ons can be saved. This path can be modified in the Add-on manager and multiple locations can be specified using the platform specific path separator (colon on Linux, semicolon on Windows). By default, a location in the user home directory is set. Add-ons discovered on the additional search path are disabled by default, and must explicitly be enabled in the Add-on manager.

If multiple add-ons with the same filename are found, only the first one on the path will be loaded. This way, a default add-on can be overridden by a custom version placed in the additional search path.

Installing Additional Add-ons

You can download more add-ons and updates from the add-on download page. To install an add-on, perform the following steps:

1. Save the downloaded Python script to one of the locations configured in the additional search path.
2. Open the Add-on Manager.
3. Find the new add-on and select its check box to enable it.

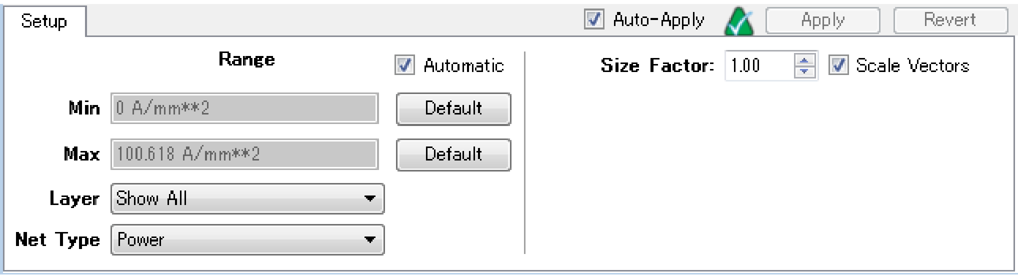
If the downloaded add-on is an update of the existing add-on, save the the update on the additional search path. Restart ADS and the updated version should be found and loaded instead of the original.

Once an add-on is added, it cannot be unloaded until the program is exited. While managing add-ons during a session, consider the following factors:

- When a loaded add-on is disabled, the status icon will still be shown in the Add-on Manager. Only the next time that ADS is started, the add-on will no longer be loaded.
- When an add-on fails to load because of a Python exception or other error, it will remain in this state even if the error is fixed. Restart ADS to load the add-on.
- When add-ons with the same file name are found on the search path, only the first to be found is loaded. However, if the first add-on was added only to the path while ADS is already running, it does not overrule an add-on by the same name that is already loaded, even if the latter one is found further on the search path.

Setup Options for Customizing Results

While viewing the results of an analysis, you can specify the setup options displayed in the following figure:



Range

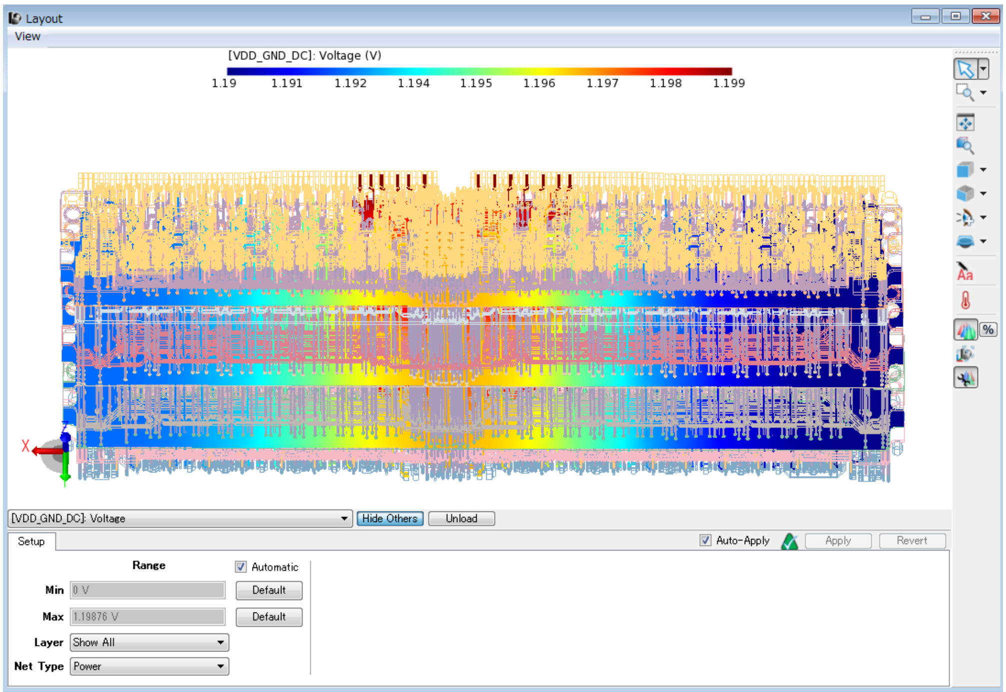
The following table describes the Range options:

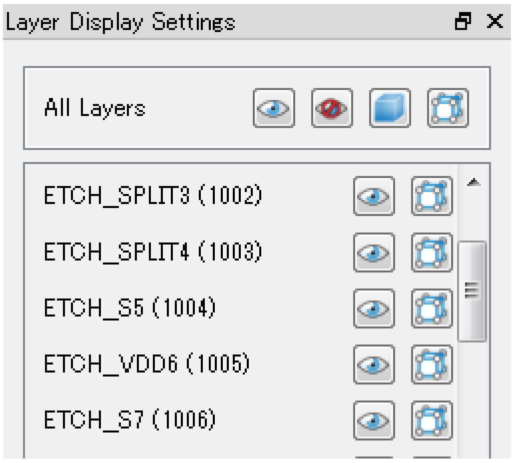
Range	Description
Min	Minimum value of the scale bar
Max	Maximum value of the scale bar
Layer	Layer name to show the values
Net Type	Net type to show the values (Not available in some type of filed plots)
Automatic	Automatically set the Min/Max values. Turn this option off if you want to specify your preferred Min/Max.

For complex/multi-layered board, it might be difficult to display the Field plots for all layers. In such a scenario, using Layer Display Settings add-on to limit the displaying layers as well as Layer selector and/or Net Type selector in the Field plot setup provides a better view.

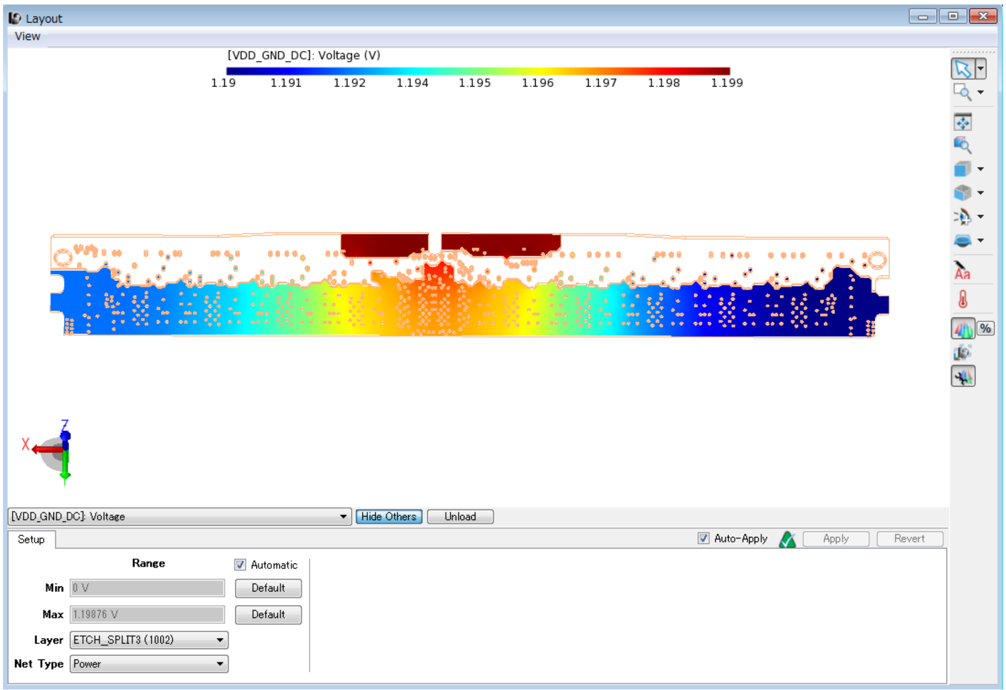
Select the following options for a better view:

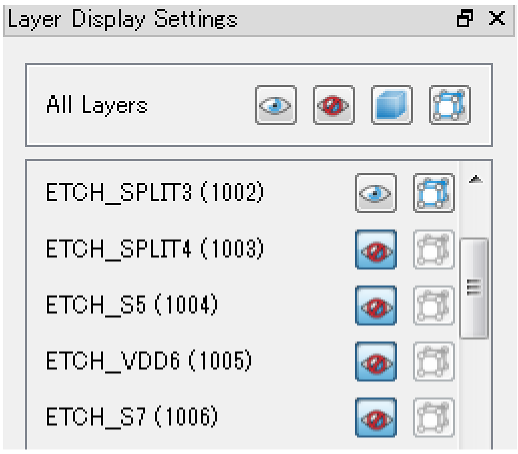
- From the **Layer** drop-down list, select **Show All**, from the **Net Type** drop-down list, select **Power** , and in the **Layer Display Settings panel,** select **All Layers Visible**,





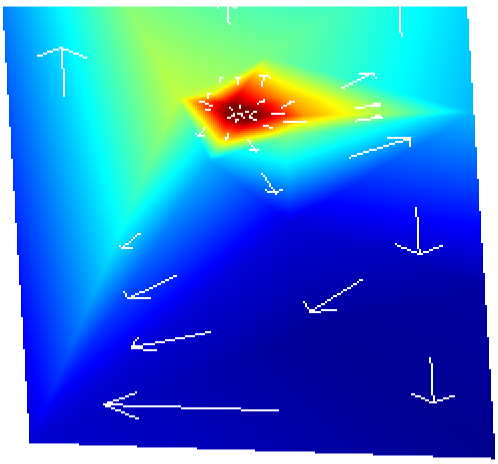
- **Layer:** ETCH_SPLIT3, **Layer Display Settings:** ETCH_SPLIT3, **Net Type:** Power

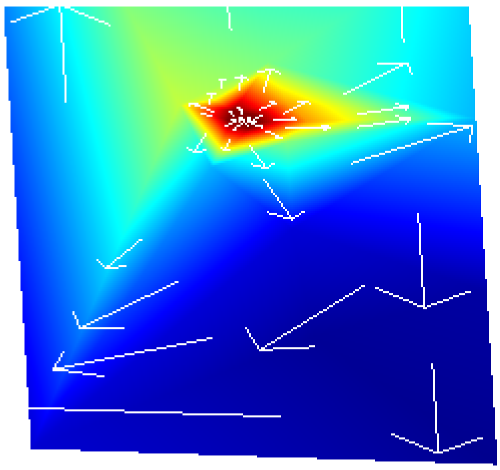




Size Factor

Specify the size factor of the arrows in the window. (Below left is Size Factor = 1, Below right is Size Factor = 2)





Excitation: Specify a excitation source from either VRMs or Sinks you defined in the setup.(Not available in some type of filed plots)

Frequency: Specify a frequency to display values.(Not available in some type of filed plots)

Hiding and Unloading Results

While viewing results in the Layout window, you can hide and unload other results.

- To hide other results, click the **Hide Others** button.
- To unload other results, click the **Unload** button.

See Also

- Viewing PI-DC Analysis Results
- Viewing PI-AC Analysis Results
- Viewing PI-PPR Analysis Results
- Viewing Power Aware SI Analysis Results

Using Component Models

Using Component Models

Contents

- Creating and Editing Component Models for Analysis
- Using Vendor Parts DB Browser

Creating and Editing Component Models for Analysis

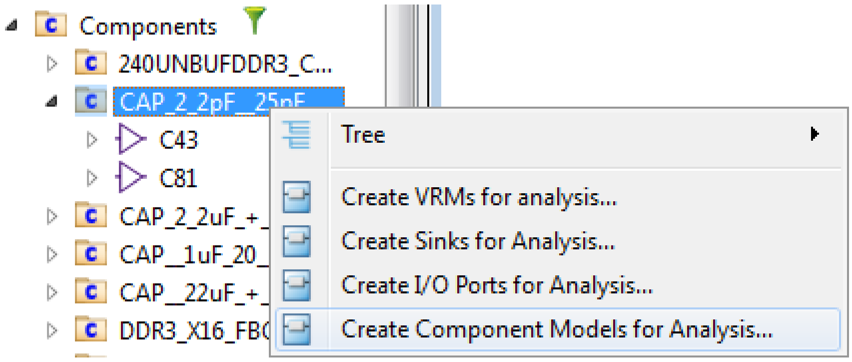
For an SIPro/PIPro analysis, you need to calculate the impact of decoupling capacitors. Identify the component models and define electrical models for the selected components by defining component models.

In this section, you will learn how to create component models and then add a lumped, SnP, or model DB for analysis.

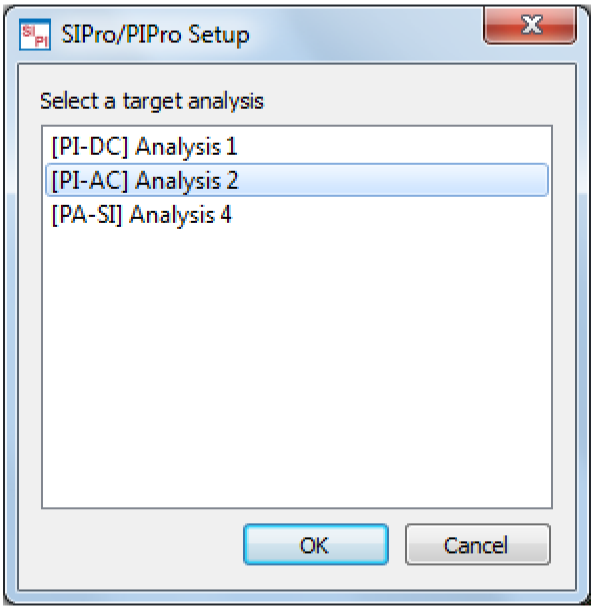
Creating Component Models for Analysis

To create a component model for analysis:

1. Select a component group in the **Components** list.
2. Right-click the selected component group and select **Create Component Model for Analysis**. The SIPro/PIPro Setup window is displayed.



3. Select the target analysis where you want to create the component model:



4. Click **OK**.

NOTE You can also drag and drop the selected components on **Component Models** in the SIPro/PIPro Setup. The instances will be grouped by component name.

Editing a Component Model for Analysis

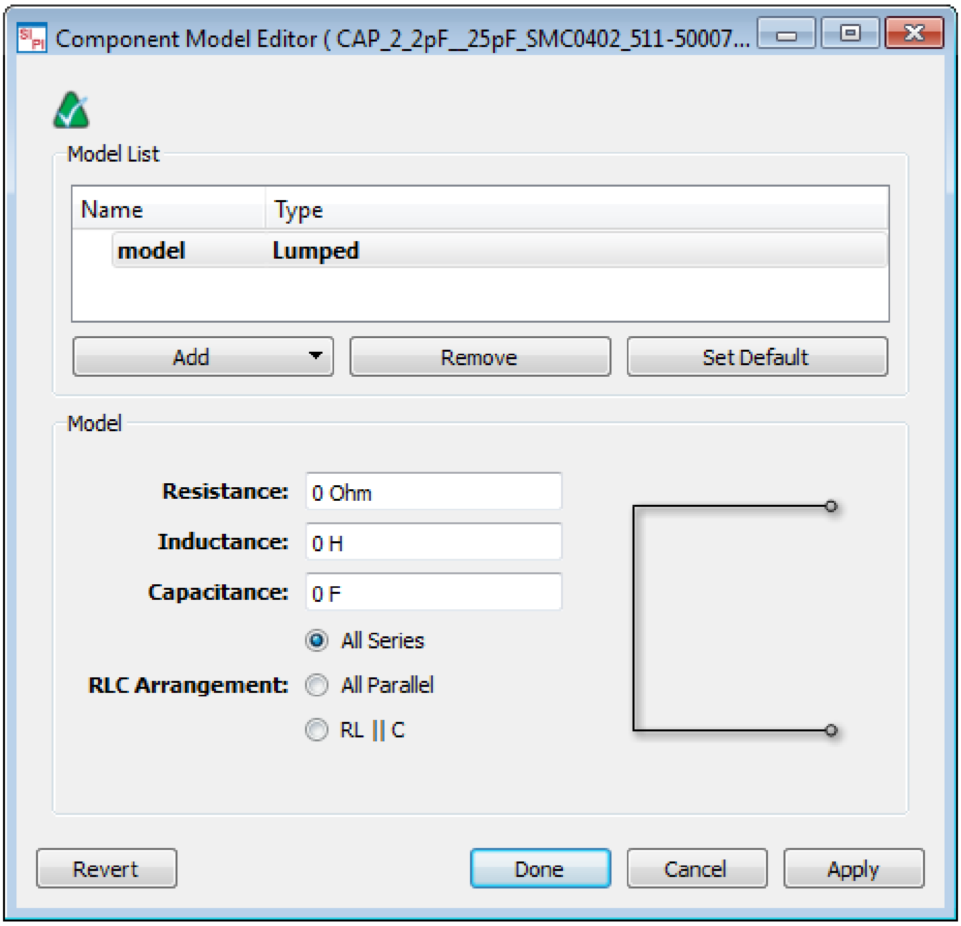
After creating a component model in the SIPro/PIPro Setup panel, you can define the electrical model parameters. The following types of models can be added:

- Lumped
- SnP
- Model DB
- Multiple Model DBs

Adding a Lumped Model

To define a lumped component model:

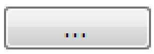
1. Double-click the component group in the SIPro/PIPro Setup panel. The **Component Model Editor** is displayed.
2. Select **Lumped** from the **Add** drop-down list.
3. Specify the **Resistance**, **Inductance**, and **Capacitance** value.
4. Select an **RLC arrangement** option.

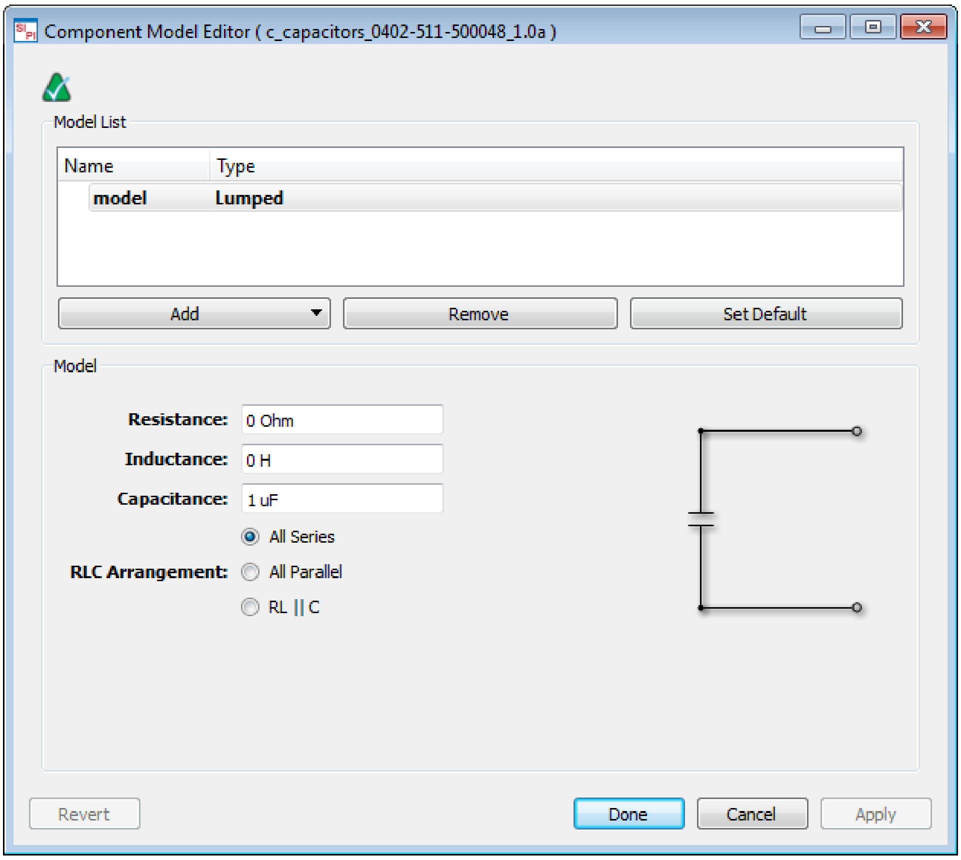


5. Click **Apply**.
6. Click **Done**.

Adding an SnP Model

You can add an SnP model for multi-port components, such as 4-pack resistors and 7-pack resistors. To define an SnP component model:

1. Double-click the component group in the SIPro/PIPro Setup panel. The **Component Model Editor** is displayed.
2. Select **SnP** from the **Add** drop-down list.
3. Specify the S parameter file in the File Name text box. You can also click  to browse to the required file.

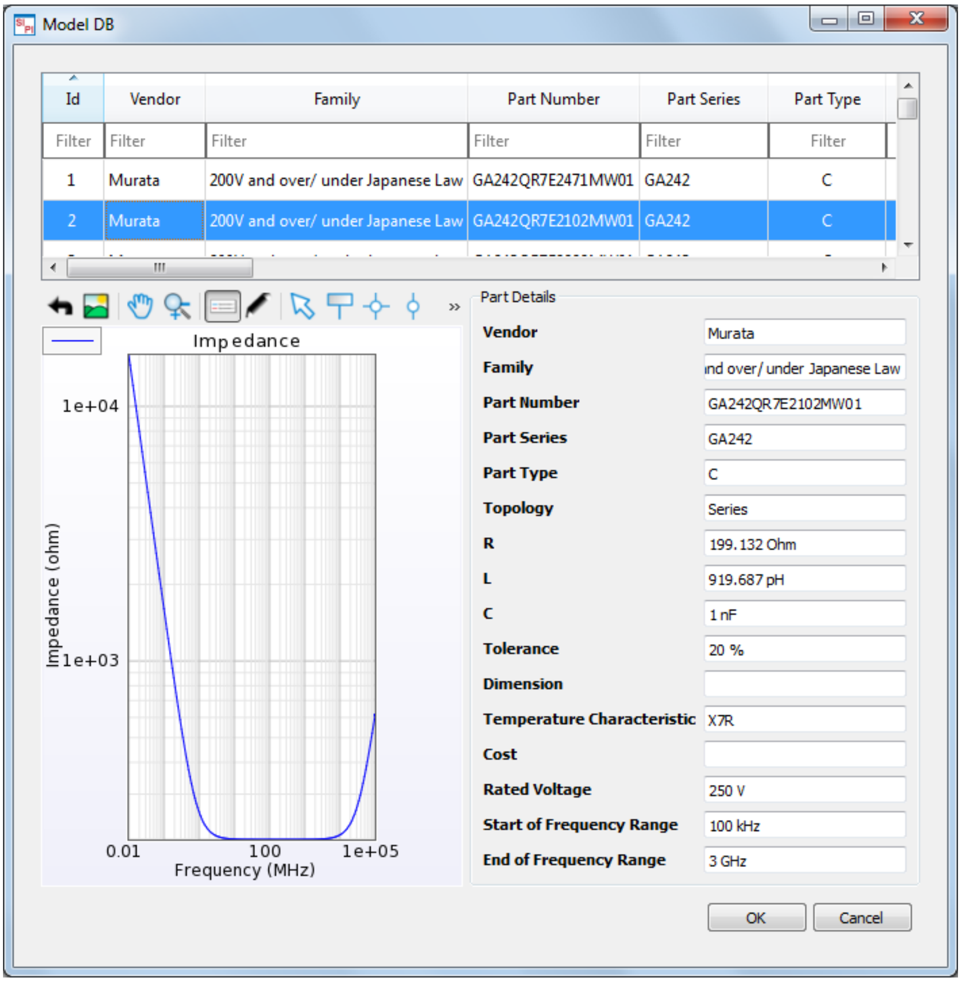


4. Click **Apply**.
5. Click **Done**.

Selecting an Existing Model DB

To use an existing model:

1. Double-click the component group in the SIPro/PIPro Setup panel. The **Component Model Editor** is displayed.
2. Select **Model DB** from the **Add** drop-down list.
3. Click  in the **Model** section.
4. The **Model DB** window is displayed.
5. Select the required model.



6. Click **OK**. All the fields are filled with the details of the selected model.

7. Click **Apply**.

8. Click **Done**.

Selecting Multiple Model DBs

To use an existing model:

1. Double-click the component group in the SIPro/PIPro Setup panel. The **Component Model Editor** is displayed.

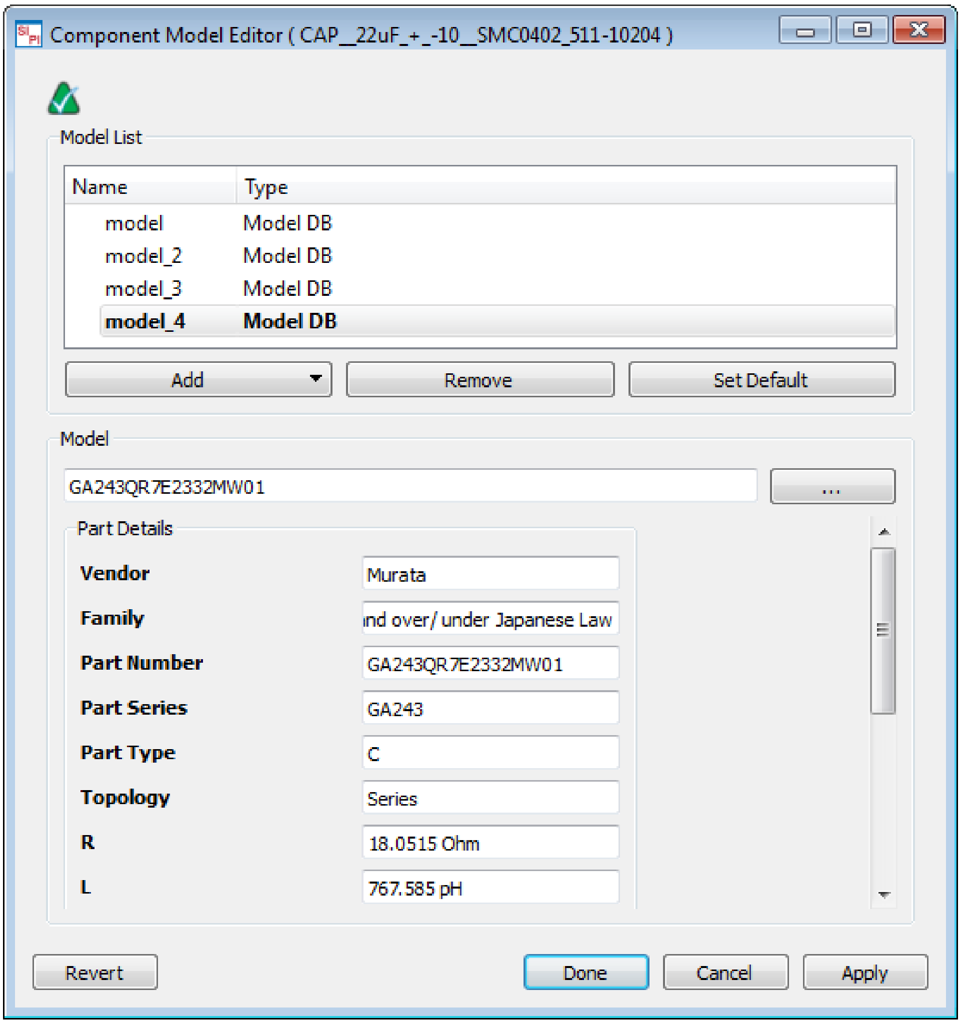
2. Select **Multiple Model DB** from the **Add** drop-down list.

3. Click ... in the **Model** section.

4. The **Model DB** window is displayed.

5. Select the required models.

6. Click **OK**. All the fields are filled with the details of the selected model.



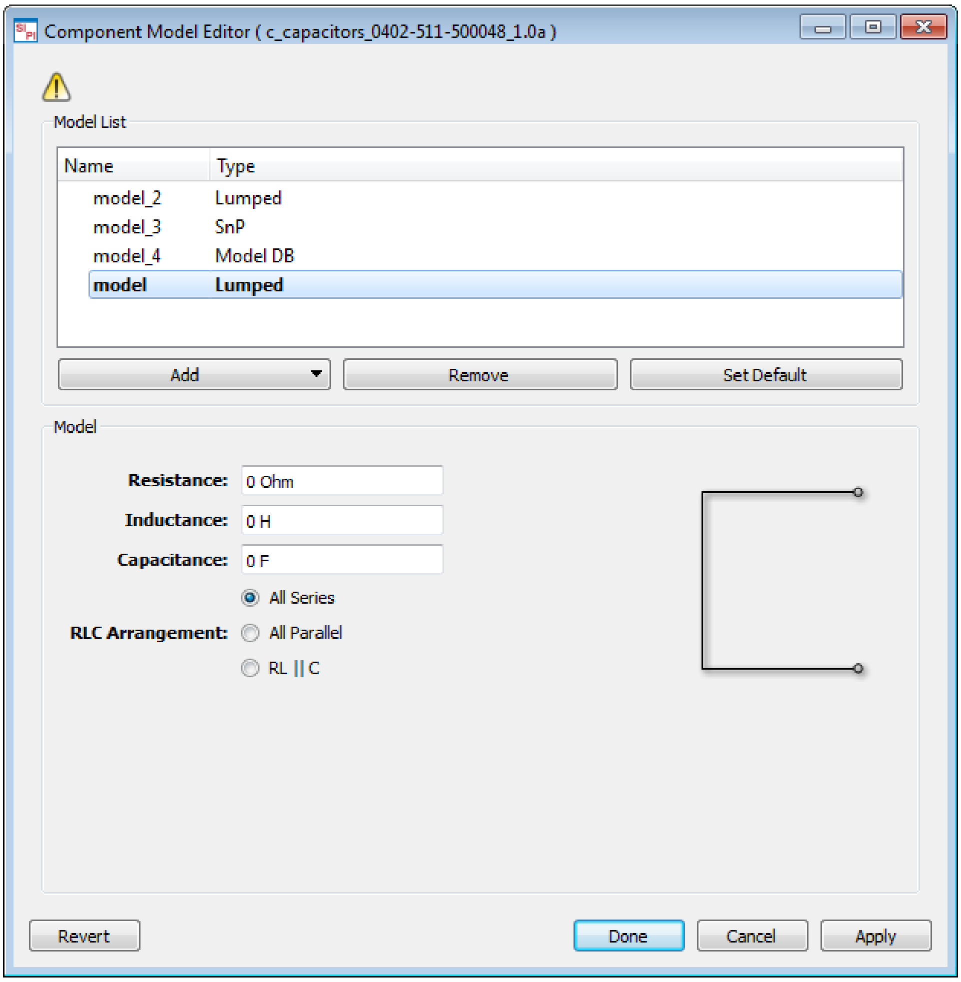
7. Click **Apply**.
8. Click **Done**.

Specifying a Default Model

If you have added multiple models, you can select a default model that will be considered during an analysis. After you have completed a simulation, you can change the default model and view results immediately. You do not need to re-simulate your design.

To specify a default model:

1. Double-click the component group in the SIPro/PIPro Setup panel. The **Component Model Editor** is displayed.
2. Select a model in the **Model List** section.
3. Click **Set Default**. For example, in the following figure, **model_4** is selected:



4. Click **Apply**.
5. Click **Done**.

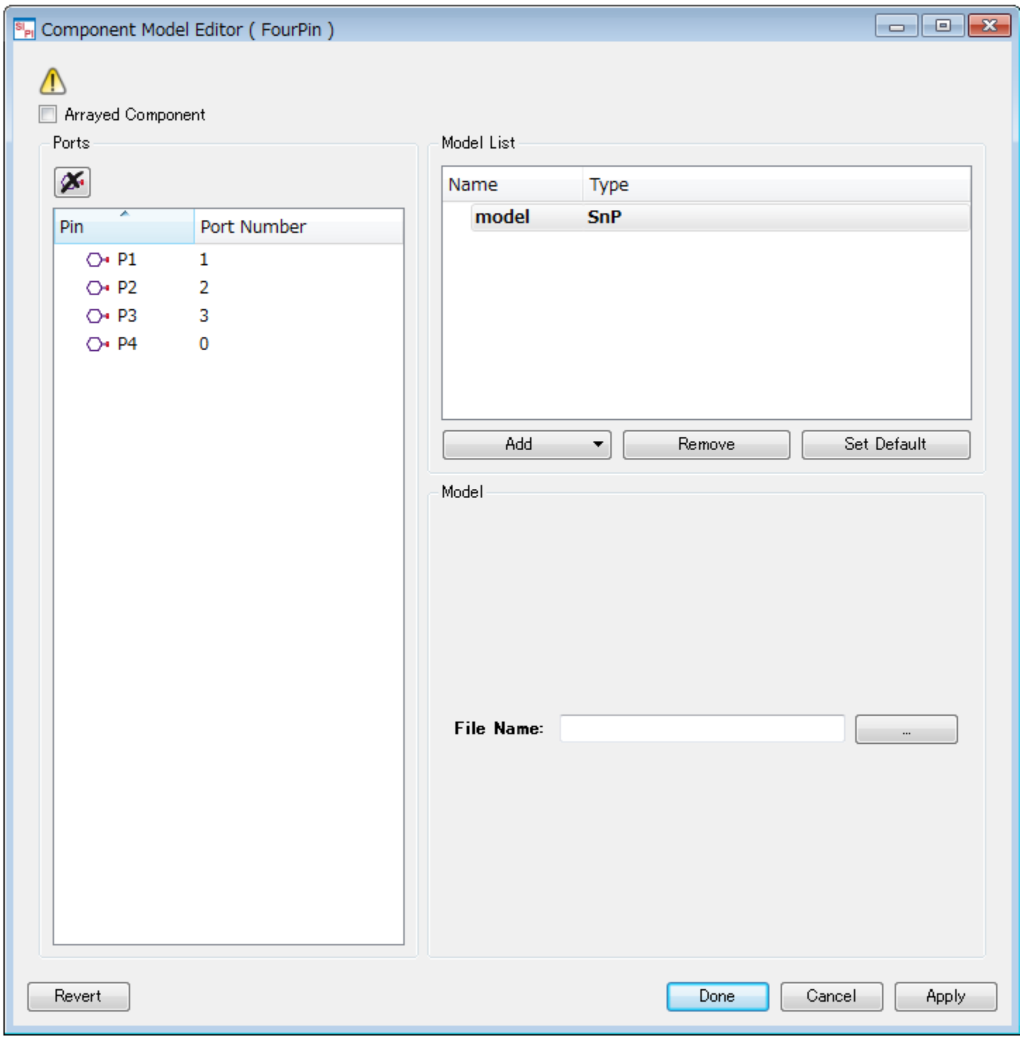
Specifying a Model with more than 2 pins

If you have a component that has more than 2 pins, you can specify port numbers for each pins to define how the model is connected when simulated.

If number of pins of component is more than 2 Ports pane appears in the left of the dialog.

To specify ports and models

1. Modify **Port Number** column so each pin connection matches the actual model's port numbers. "0" means common reference ground. In below example, 3 ports are defined where P1(+) and P4 (-) as Port1, P2(+) and P4(-) as Port2, P3(+) and P4(-) as Port3. You can make Port Number column blank if you don't want to connect model to that pin.
2. Specify model(s) that matches the number of ports you defined in Ports pane.

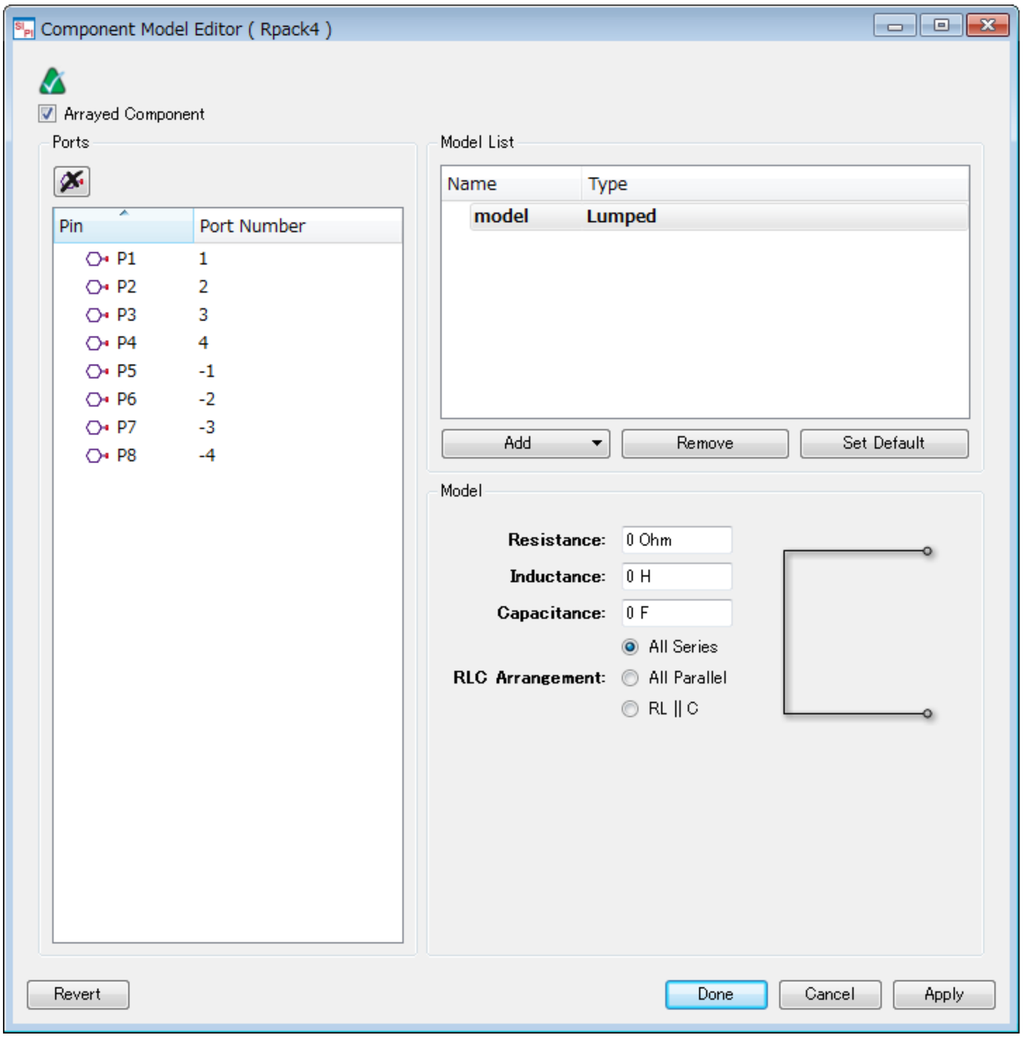


Specifying a Arrayed Component

If you have an arrayed component contains multiple 2-pin components, you can define it by using "Arrayed Component" checkbox.

To specify a Arrayed Component

1. Check **Arrayed Component** checkbox. Note that the checkbox only appears when the component has more than 2 pins.
2. Specify Port Numbers for each pins. Use "-" sign to make pin connects to minus terminal of the port. In below example, P1 connects to plus terminal of port1 and P5 connects to minus terminal of the port1.
3. Specify 2-pin model(s). The same model will be applied to each ports.



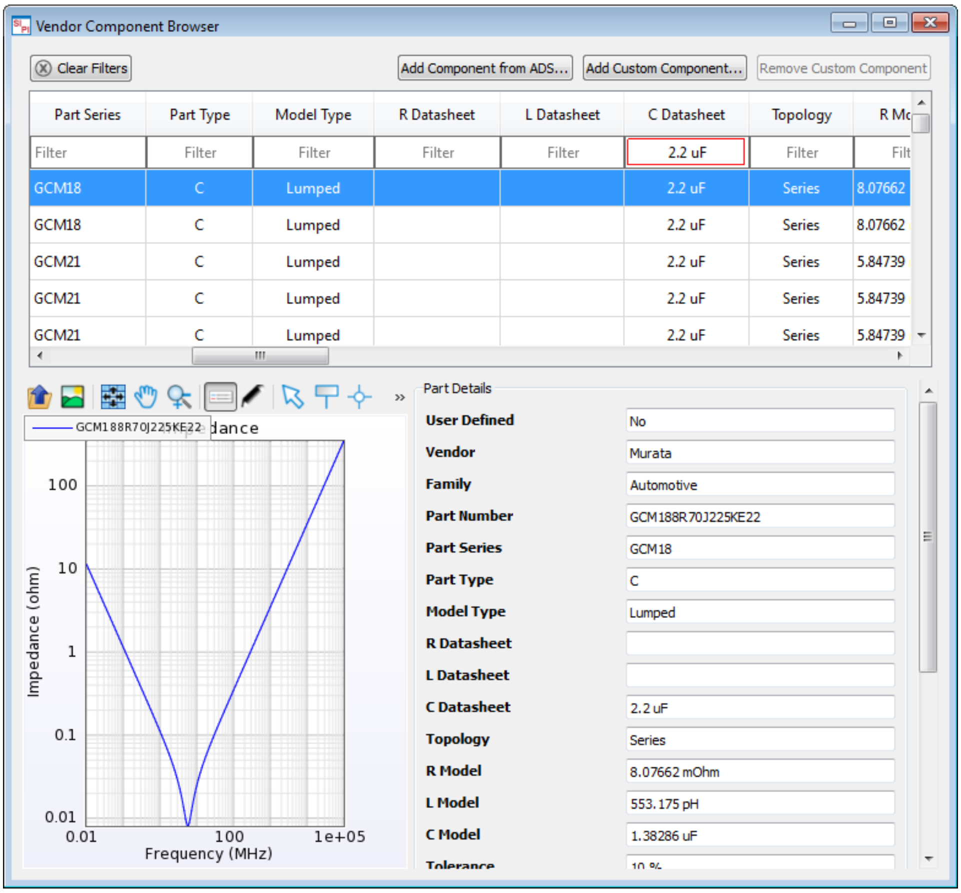
Using Vendor Parts DB Browser

Using Vendor Parts DB Browser

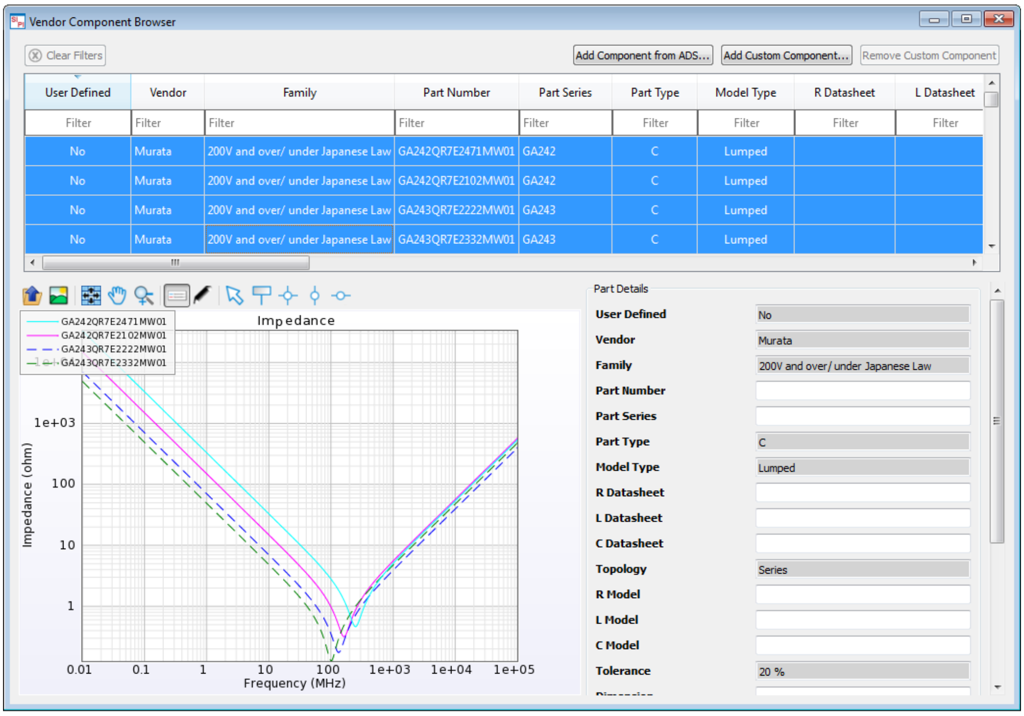
SIPro/PIPro provides a database of vendor part electrical models. It provides a complete set of MuRata, Samsung, and TDK components. You can select an electrical model for the required capacitor in the Vendor Parts DB Browser window.

To display the Vendor Parts DB Browser:

1. Select **Tools** > **Vendor Parts DB**. The Vendor Parts DB Browser window is displayed.
2. Select the required electrical model. You can also use the **Filter** text field to search the required model. Type a value in the required column. You can use multiple filter values. For example, the following figure displays results of the **2.2uF** capacitors search.



In the Vendor Parts DB Browser window, you can also view the impedance of various capacitors. You can select multiple capacitors in the window and compare the impedance value, as shown in the following figure:



The properties that are the same for all selected components appear grayed out in the Part Details panel.

Adding Components to the Vendor Parts DB

SIPro/PIPro allows you to extend the existing vendor parts database. The following types of models are supported:

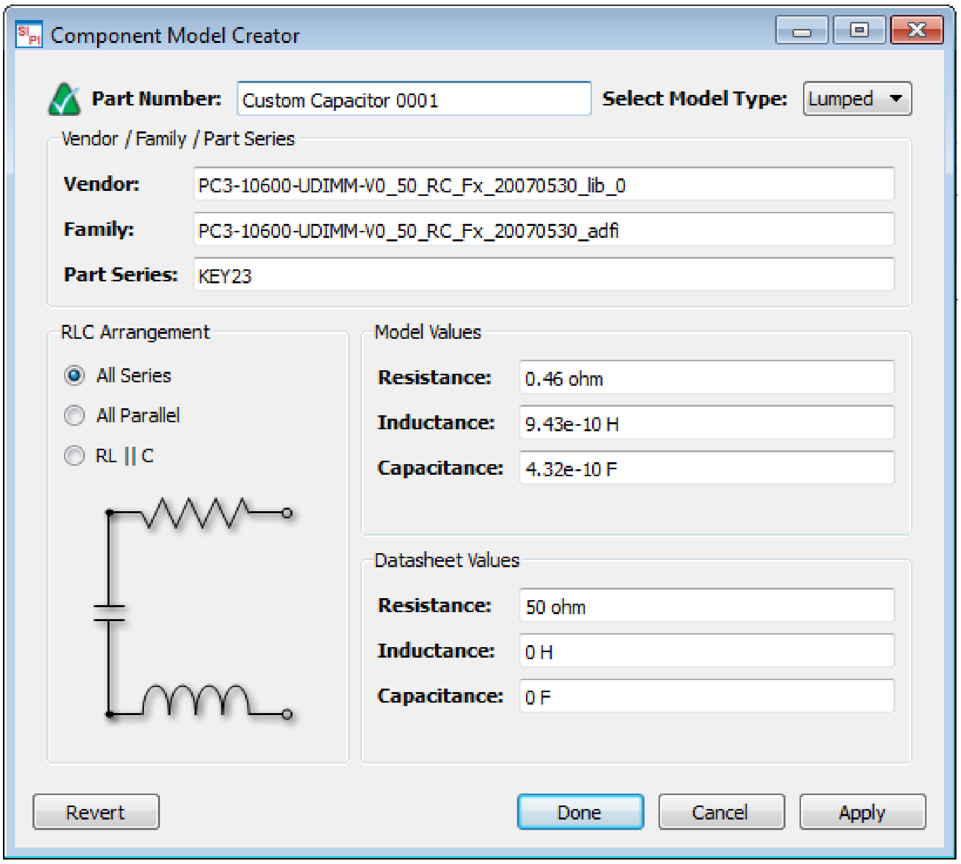
- Component from ADS
- Lumped
- SnP

Adding Custom Component

Adding a Lumped Model

To add a custom lumped model type to the Vendor Parts DB:

1. Click the **Add Custom Component** button in the *Vendor Components Browser* window to open the Component Model Creator dialog box.
2. Select the **Lumped** model type from the **Select Model Type** drop-down list.
3. Specify the Lumped Model properties:

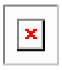


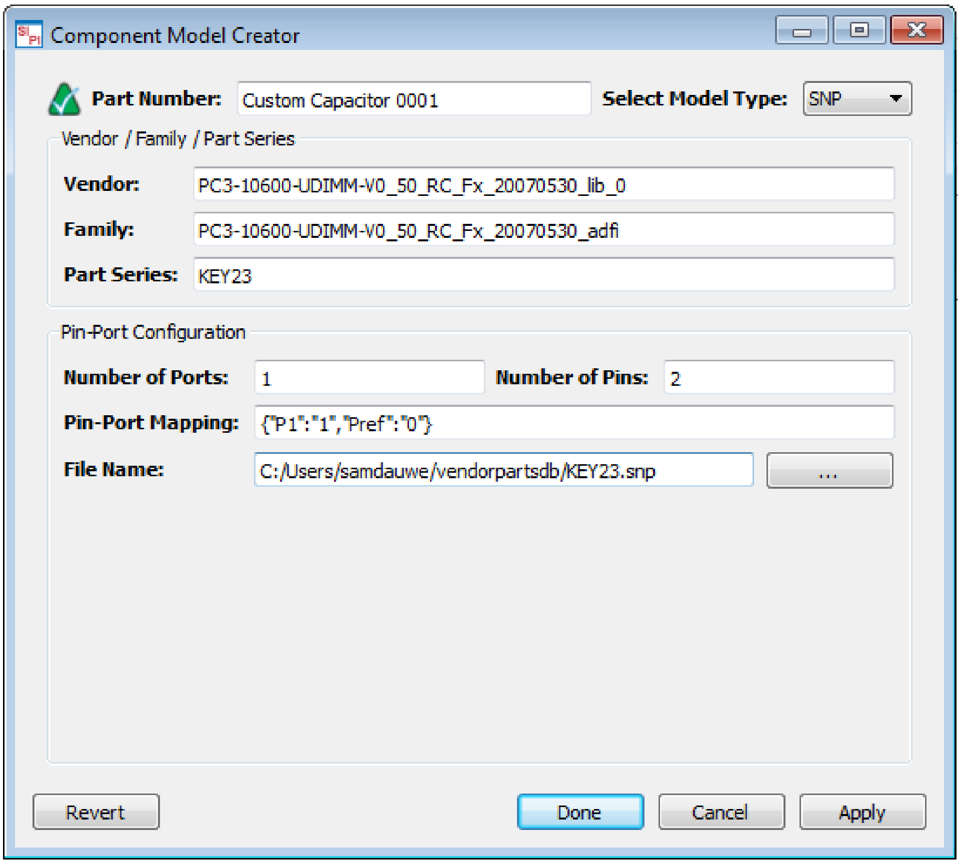
4. Click **Apply**.
5. Click **Done**.

Adding an SnP Model

To add a custom SnP model type to the Vendor Parts DB:

1. Click the "**Add Custom Component...**" button in the *Vendor Components Browser* window to open the Component Model Creator dialog.
2. Select the **SnP** model type from the **Select Model Type** drop-down list.

3. Specify the S parameter file in the File Name text box. You can also click  to browse to the required file. The pin-port configuration is automatically generated from the S parameter file.



4. Click **Apply**.
5. Click **Done**.

PIPro Analysis

PIPro Analysis

Contents

- Tutorial-Performing a PI-DC and PI-AC Analysis
- PIPro Analysis Setup Overview
- PI-DC Static IR Drop Analysis
- PI-AC Dynamic IR Drop Analysis
- PI-PPR Analysis

Tutorial-Performing a PI-DC and PI-AC Analysis

In this tutorial, you will learn how to create a PI-DC and PI-AC analysis setup, run the analysis, and view results. The example workspace contains a Samsung DDR4 UDIMM memory card. The design files are downloaded from JEDEC (www.jedec.com) and used for demonstration purposes. The example design consists of a 6 layer board with a single power rail for core and I/O buffers, as shown in the following figure:

You can view the nets, components instances, and individual nets in the navigator window. The main power net passes from the conductor at the bottom of the design to the devices at the top. GND nets are present on the design. The substrate pack consists of routing on the top and bottom layer. The power and GND nets are on layer 3 and 4. Before performing the PIPro analysis, you need to know the names of the power and GND nets.

In this example design, the VRM is connector J1, which is the source of power supply. It is the connector device at the bottom of DDR4 board. There are eight sinks, which are the ICs distributed over the board. The power supply is consumed by sinks. You will analyze the impact of decoupling capacitors.

You can access the example workspace from the following location:

examples/HSD/SIPro_PIPro_Getting_Started_Example_wrk.7zadsfcv

In this tutorial, you will perform the following tasks:

- Perform a PI-DC analysis
- Perform a PI-AC analysis

Performing a PI-DC Analysis

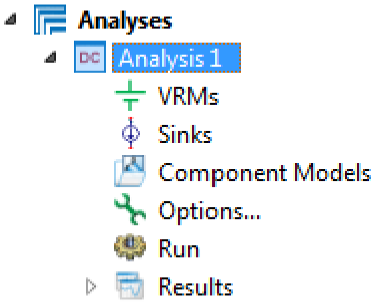
You can analyze the voltage drop in the VDD-GND power distribution network. The supply power comes in through connector 'J1', the ICs that consume current are the 'DDR4...' component instances. There are no series components.

To perform an analysis, open the SIPro/PIPro Setup window:

1. Open a Layout window in ADS.
2. Select **Tools > SIPro/PIPro > Open Setup** from a Layout window to create a new or open an existing setup. The SIPro/PIPro Setup window is displayed.

Creating a PI-DC Analysis Setup

To create a PIPro Static IR Drop (PI-DC) analysis, use the default empty template available in the project panel. Expand **DC Analysis 1** in the SIPro/PIPro panel, as shown in the following figure:



To create a setup, you need to define the following:

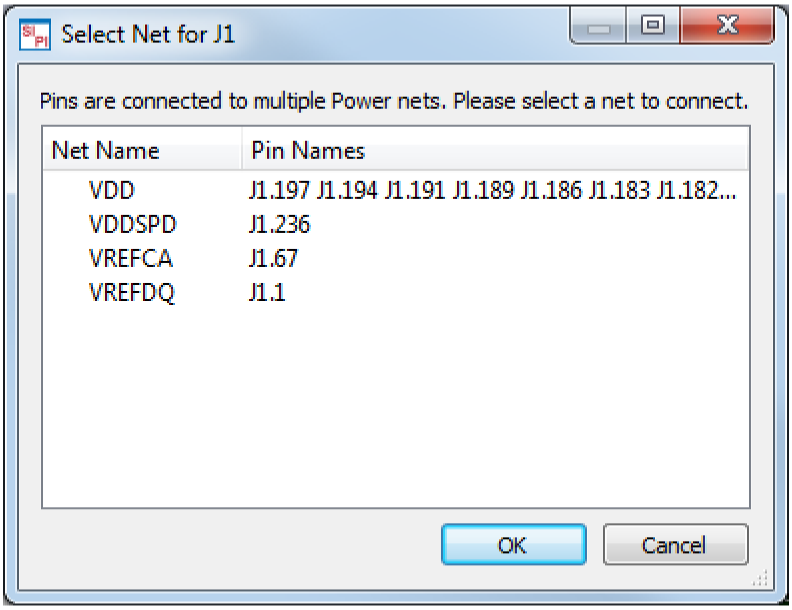
- VRM
- Sink
- Options

Defining a VRM

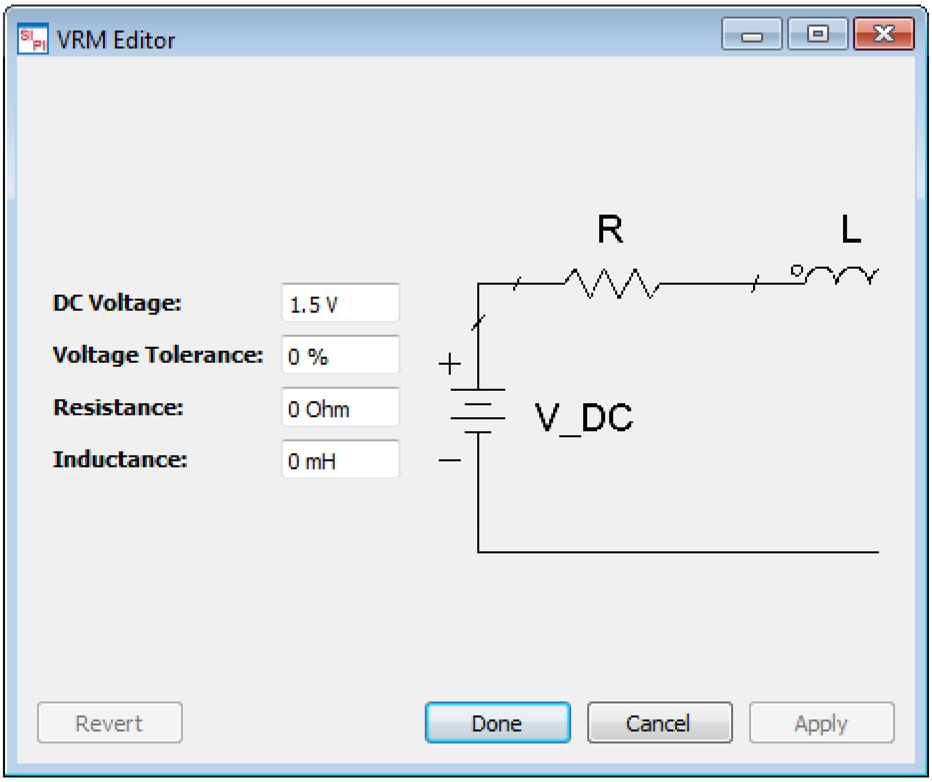
To setup a VRM:

1. Select the **J1** connector component instance in the **Components** list, as shown in the following figure:

2. Drag and drop the selected instance on the **VRMs** part in the SIPro/PIPro Setup panel. The Select Net for J1 window is displayed.



3. Select **VDD**.
4. Click **OK**. A new VRM definition is added in the SIPro/PIPro Setup panel.
5. Double click the VRM definition to open the **VRM Editor** dialog box.

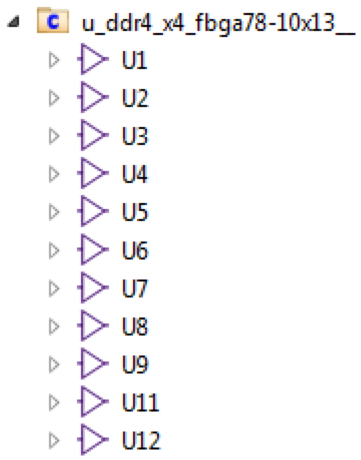


6. Specify the **DC Voltage**, **Voltage Tolerance**, **Resistance** and **Inductance**.

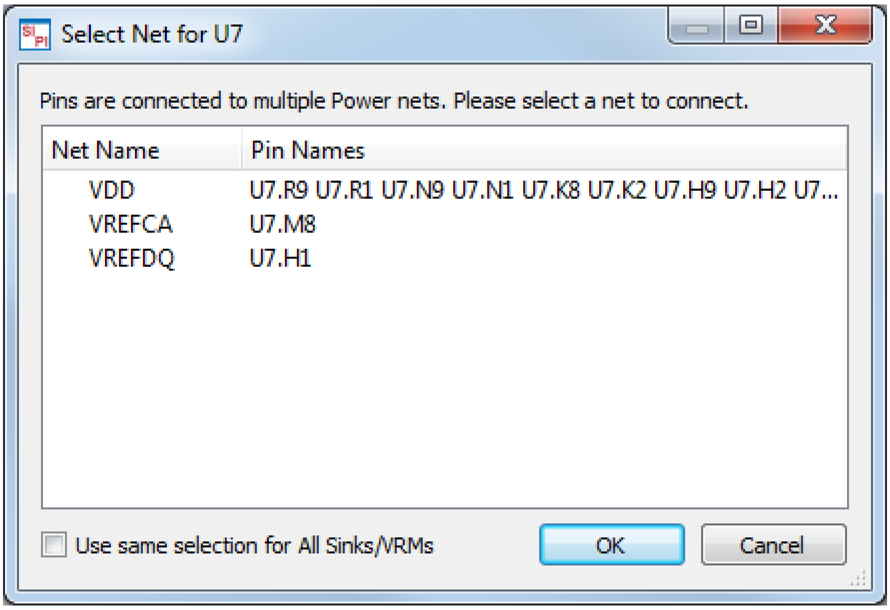
Defining a Sink

Sinks are devices that consume current. To define a sink:

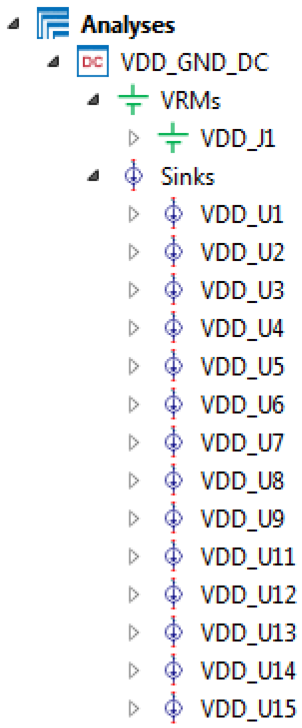
1. Select **u_ddr4_x4_fbga78-10x13** at component level in the **Components** list, as shown in the following figure:



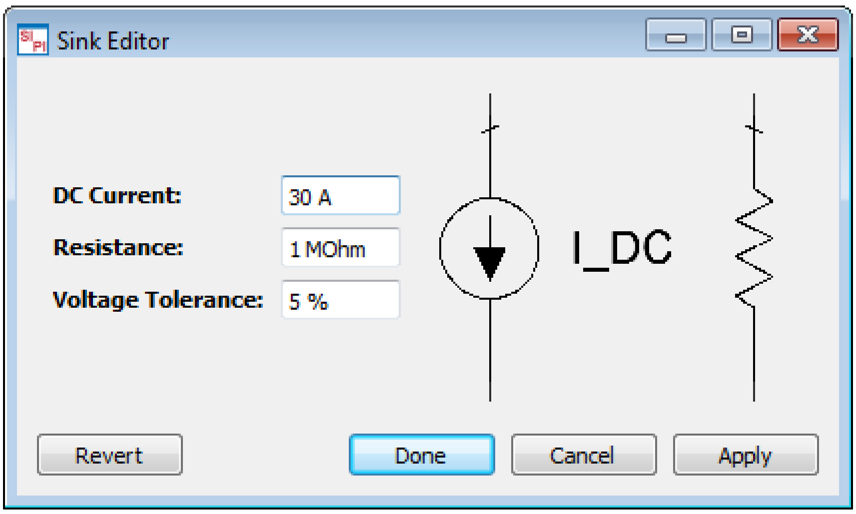
2. Drag and drop the component on **Sinks**. Since the component connects to multiple power nets, the **Select Net for U7** dialog box is displayed.



3. Select **VDD** as the power net you want to analyze.
4. Select the **Use same selection for All Sinks/VRMs** option.
5. Click **OK**. A Sinks definition is added having a list of 8 sinks, as shown in the following figure:



6. Double click **sink_U4** in the **Sinks** list to open the Sink Editor dialog box.
7. Type **30** in the **DC Current** field.



8. Click **Done**.

Defining Options

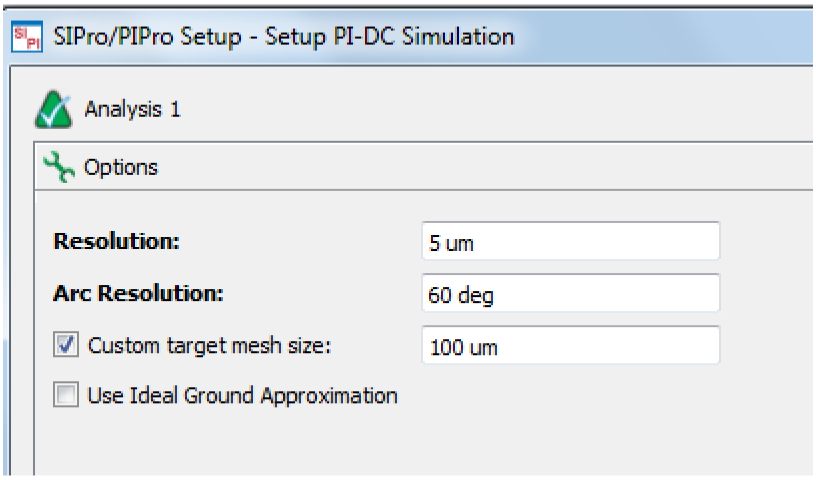
A PI-DC analysis provides the following options:

- Use Ideal Ground Approximation. When enabled, this option will treat all Ground nets as ideal shorts. The power rail is often the dominant contributor to the voltage drop. This option provides you the capability to analyze this dominant factor in less simulation time.

To specify options:

1. Double-click **Options** in the Analyses list. The Setup PI-DC Simulation window is displayed.
2. Select **Use Ideal Ground Approximation**.

3. Select **Custom target mesh size** and specify the required value.



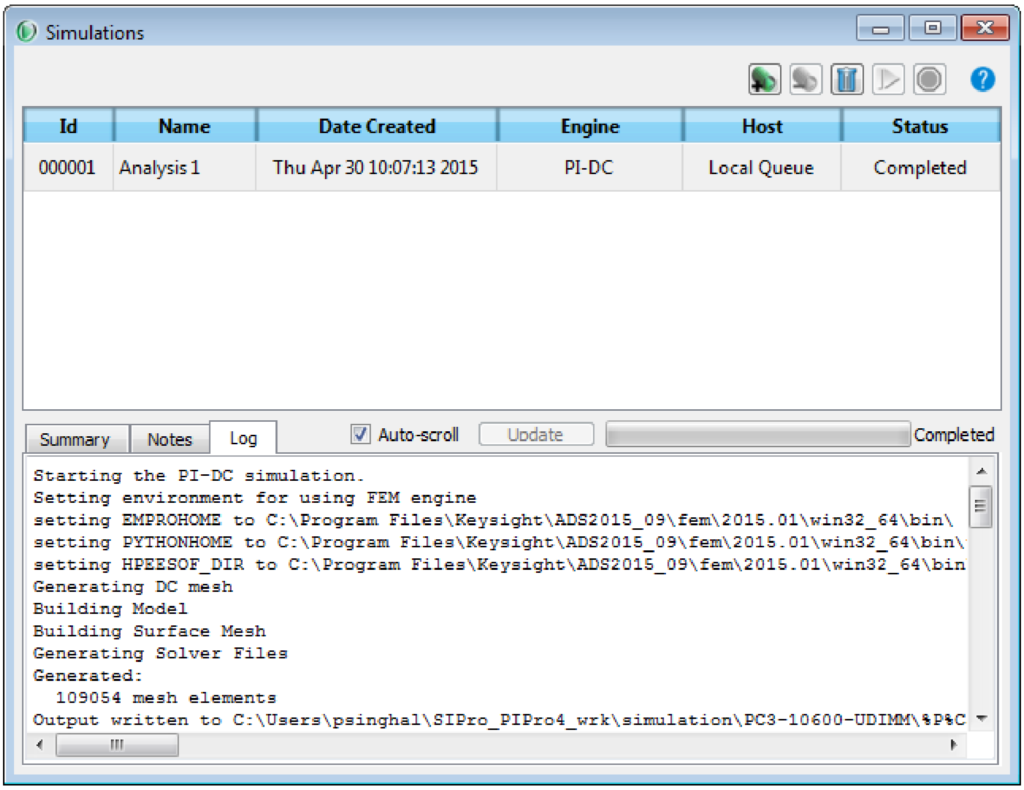
4. Click **Done**.

Saving the Analysis Setup

After defining your analysis setup, select **File** > **Save**. Now, you are ready to perform the analysis.

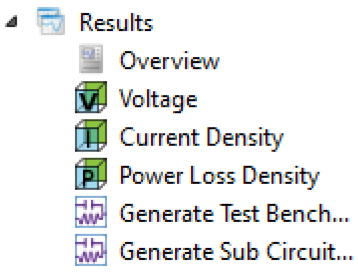
Running the Analysis

Double-click **Run** to start the PI-DC analysis. The **Simulations** window is displayed and the PI-DC analysis is initiated.



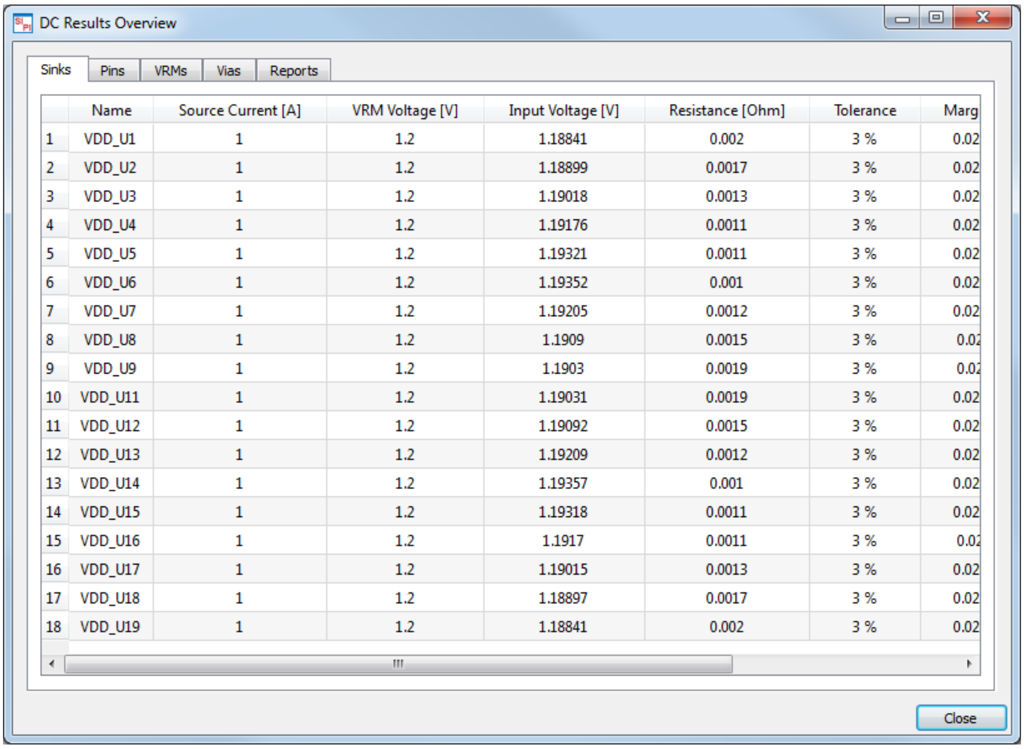
Viewing Results

After the simulation is complete, the results are listed in the project panel. You can view the results by double-clicking the required Results option:



Overview

Clicking the **Overview** option opens a window with a table based overview of voltages and currents at Sinks, Pins, VRMs, and Vias.



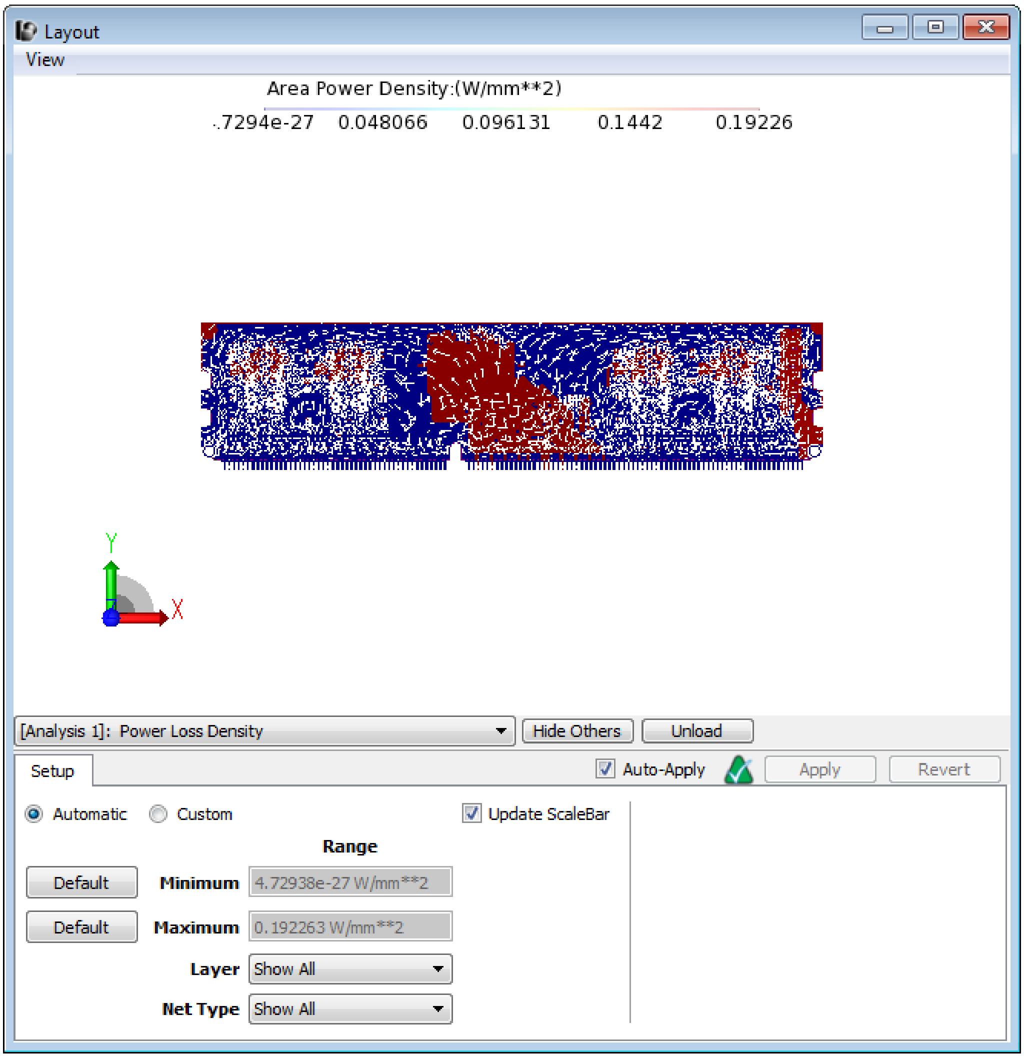
	Name	Source Current [A]	VRM Voltage [V]	Input Voltage [V]	Resistance [Ohm]	Tolerance	Marg
1	VDD_U1	1	1.2	1.18841	0.002	3 %	0.02
2	VDD_U2	1	1.2	1.18899	0.0017	3 %	0.02
3	VDD_U3	1	1.2	1.19018	0.0013	3 %	0.02
4	VDD_U4	1	1.2	1.19176	0.0011	3 %	0.02
5	VDD_U5	1	1.2	1.19321	0.0011	3 %	0.02
6	VDD_U6	1	1.2	1.19352	0.001	3 %	0.02
7	VDD_U7	1	1.2	1.19205	0.0012	3 %	0.02
8	VDD_U8	1	1.2	1.1909	0.0015	3 %	0.0
9	VDD_U9	1	1.2	1.1903	0.0019	3 %	0.0
10	VDD_U11	1	1.2	1.19031	0.0019	3 %	0.02
11	VDD_U12	1	1.2	1.19092	0.0015	3 %	0.02
12	VDD_U13	1	1.2	1.19209	0.0012	3 %	0.02
13	VDD_U14	1	1.2	1.19357	0.001	3 %	0.02
14	VDD_U15	1	1.2	1.19318	0.0011	3 %	0.02
15	VDD_U16	1	1.2	1.1917	0.0011	3 %	0.0
16	VDD_U17	1	1.2	1.19015	0.0013	3 %	0.02
17	VDD_U18	1	1.2	1.18897	0.0017	3 %	0.02
18	VDD_U19	1	1.2	1.18841	0.002	3 %	0.02

Voltage

Clicking the **Voltage** option in the Results list loads the voltage plot in the Geometry window. This is a scalar plot. The voltage is forced at 0 at the negative pins of the VRMs.

Current Density

Clicking the **Current Density** option in the Results list loads the current density plot in the Geometry window. This is a vector plot. The color indicates the current density amplitude. The white arrow indicates the direction of the current. The arrow size is not an indication for the current density amplitude.



Power Loss Density

Clicking the **Power Loss Density** option in the Results list loads the power loss density. This is a scalar plot. This plot shows the power loss per unit area.

Generate Schematic

Clicking the **Generate Schematic** option opens a Schematic window with the VRMs, Sinks, Component Models and an SNP that points to the DC S-parameter model.

Performing a PI-AC Analysis

After running a PI-DC analysis, you can perform the PI-AC analysis. A PI-AC analysis enables you to calculate the impedance of the power distribution networks from different sinks. It helps you to determine and minimize the impedance between power and ground pins at various IC locations on the board. A PDN should have the lowest impedance for achieving good power integrity. Therefore, you need to minimize the impedance between power and ground pins, which will vary based on the location of capacitors and the type and value of capacitor. The goal of PI-AC analysis is to provide low impedance to all sinks.

In this example, the impact of components placed between the capacitors is calculated. You will use component models of the MuRata library.

Enable MuRata Library

In this example, you will run simulations with MuRata library components. Download the MuRata library from the Web:

http://www.murata.com/en-global/tool/library/keysight3

Include the MuRata library in the current ADS workspace and enable this library in ADS.

Opening the Example Design

Open the *SIPro_PIPro_Getting_Started_Example_wrk.7zadsfcv* design. The PI analysis setup data for a specific design is stored in a cell view of the “SIPro/PIPro Setup”.

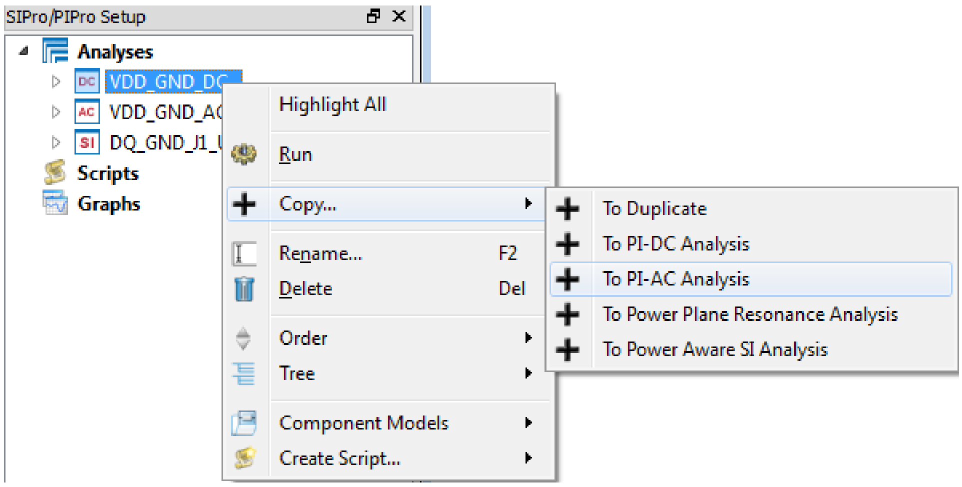
Double-click the **sipiSetup** view in the ADS Main window.

The SIPro/PIPro Setup window is displayed.

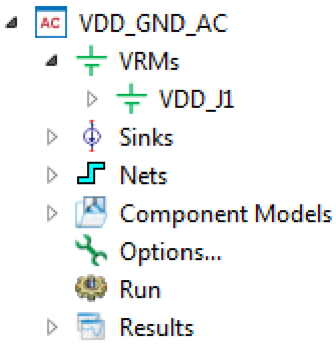
Creating a PI-AC Analysis Setup

In this example, you will copy the PI-DC analysis setup. Therefore, you do not need to redefine the VRMs, which are the connectors for supplying power, and sinks that consume the current.

1. Right-click the PI-DC analysis setup in the project panel:
2. Select **Copy** > **To PI-AC Analysis**:



3. The PI-DC analysis setup is copied to create a new PI-AC Analysis.
4. Right-click the copied analysis and select **Rename**.
5. Type **VDD_GND_AC**, as shown in the following figure:

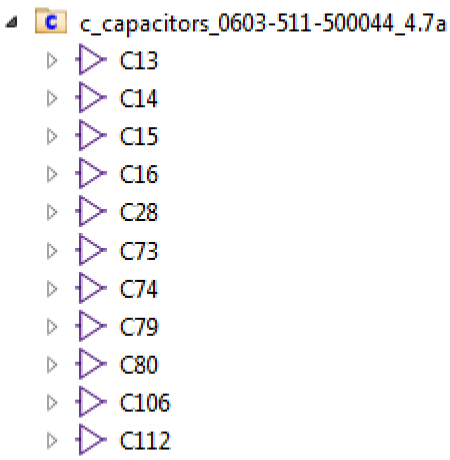


Defining a Component Model

For a PI-AC analysis, you need to calculate the impact of decoupling capacitors. Identify the component models and define electrical models for the selected components.

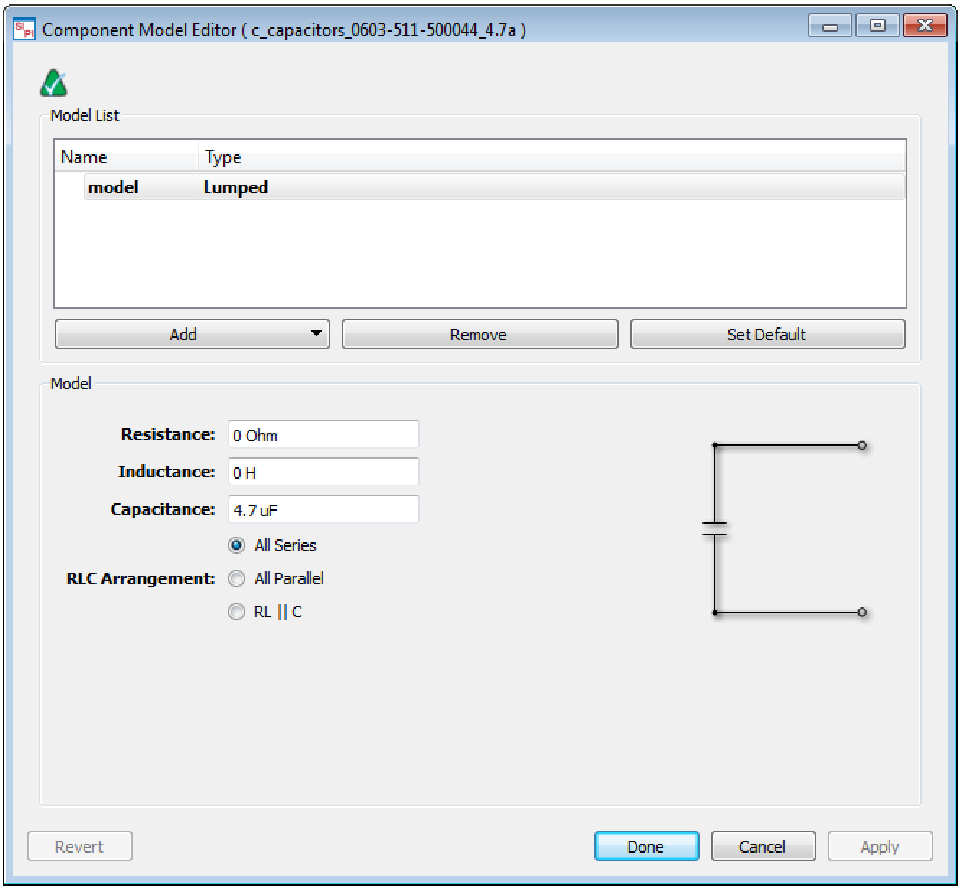
To define a component model:

1. Select the component group **c_capacitors_0603-511-500044_4.7a** in the **Components** list. This group of components will be analyzed.
2. Drag and drop the selected component on **Component Models** in the SIPro/PIPro Setup window. A new list is created, as shown in the following figure:



NOTE After adding a component model in in the SIPro/PIPro setup panel, you may need to delete the nets that are not connected to GND and VDD.

3. Double-click **c_capacitors_0603-511-500044_4.7a** in the Component Models branch to specify the electrical models to be used for analysis.
4. Select **Lumped** from the **Add** drop-down list.
5. Type 4.7 uF in the **Capacitance** field.



6. Click **Done**

NOTE

You can also search an electrical model in the database provided by the vendor. For more information, see Using Vendor Parts DB Browser and Creating and Editing Component Models for Analysis.

Defining Options

To define options:

1. Double-click **Options** in the PI-AC Analysis list in the SIPro/PIPro Setup panel.
2. Accept the default **Resolution** and **Arc Resolution** value.
3. Click the **Frequency Plans** tab.

	Type	Start	Stop	Points	Step
☑	Logarithmic	10 kHz	300 MHz	5/decade (24 points)	-
☑	Adaptive	10 kHz	300 MHz	300 (max)	-

4. Click **Done**.

Running the Analysis

Save your PI-AC analysis setup and double-click **Run** to start the PI-AC analysis. The **Simulations** window is displayed and the PI-AC analysis is initiated.

Viewing PDN Impedance

To view PDN impedance results:

1. In the **Results** list, double-click **PDN Impedance**. The PDN impedance window is displayed.
2. Select the required sinks.
3. Right-click the selected sinks and choose **Add PDN Impedance**. The PDN Impedance plot is displayed.

PIPro Analysis Setup Overview

To perform a power integrity (PIPro) simulation, you require detailed information about the power distribution network (PDN) in a design. This information includes identifying the voltage regulator module (VRM), the IC that consumes power, the components and component models, and stimulus for the analysis.

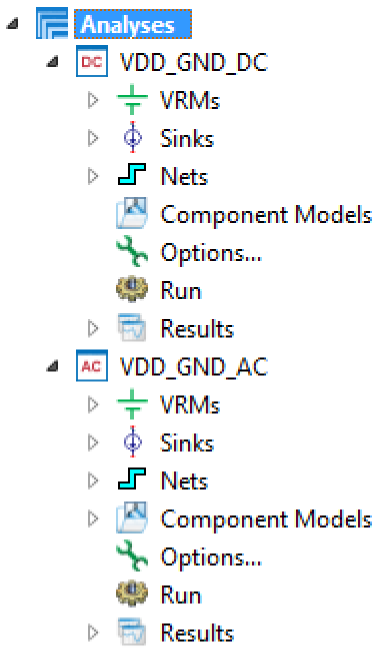
Prerequisites

Before performing a PIPro analysis, you need to gather the following Information about a PDN:

- Identify the power and ground rail net name(s) that you want to analyze and obtain the DC voltage. A power rail can consist of multiple nets that are connected through series components.
- Identify the instance name (or reference designator) of the Voltage Regulator Module (VRM). In case the VRM is not mounted on the Printed Circuit Board (PCB) and the supply power comes in through one (or more) connectors, identify these instances name(s). Also, the power rail and ground rail might come in through a separate connector. For such designs, you need to identify both connector instance names.
- Identify the instance name(s) of the ICs that consume power. Obtain the DC current consumption properties of each IC. This is the total current that will be distributed among the set of individual pins of the IC. Obtain the allowed DC voltage drop for each power rail. This is the recommended minimum DC supply voltage that is needed by the IC to guarantee proper operation. You might consult the datasheet, ask the vendor, run the power calculator provided by FPGA vendors, or ask the IC designers.
- Identify the instance name(s) of all series components on the power rail. These are typically resistors, inductors, or fuses. Obtain the electrical model parameters for them (RLC values and topology).

Setup

In a new SIPro/PIPro setup, you can use the empty templates for PI-DC, PI-AC, and PPR analysis in the Project panel. You can also create a new analysis. To create a new analysis, right-click **Analyses** and choose the required analysis. The following figure displays the PI-DC and PI-AC setup:



VRMs

A voltage regulator module (VRM) allows processors with different supply voltage to be mounted on the same motherboard. A VRM regulates the voltage fed to a microprocessor. While creating an analysis setup, you can define various properties of a VRM, such as the DC voltage, voltage tolerance, and resistance.

Sinks

A sink is a component, such as ICs, that consumes electricity in a design. While creating a setup, you can define the amount of current the ICs will consume by defining the properties of a sink, such as the DC current, resistance, and voltage tolerance.

Options

The following table describes various options available for a PIPro analysis:

Option	Description
Resolution	This option specifies that points closer than the Resolution distance will be considered as equal during simulation
Arc Resolution	Arc resolution is used to discretize arcs and circles during simulation
Custom Target Mesh Size	Specifies the mesh size for generating the global 3D mesh during simulation
Use Ideal Ground Approximation	Ground net metalization is modeled as a perfect conductor during simulation

PI-DC Static IR Drop Analysis

PI-DC Static IR Drop Analysis

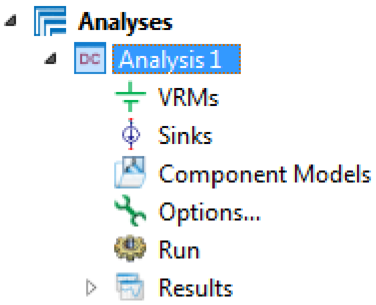
- Creating a PI-DC Analysis Setup
- Viewing PI-DC Analysis Results

Creating a PI-DC Analysis Set up

Using SIPro/PIPro, you can perform a PI-DC analysis to compute the voltage, IR drop (voltage drop), current, and power loss density in the power supply nets. The DC analysis helps you to identify the IC and connector pins or stitching vias drawing large amounts of current at DC.

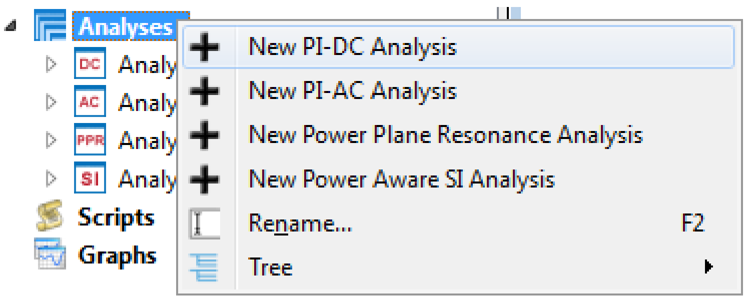
Due to excessive voltage drop, the power supply voltage at the IC might fall below the recommended minimum voltage. This can cause malfunctioning of the IC. Excessive current density in the perforated power supply rails can generate excessive heat, which might lead to board failures due to de-lamination or fusing. Also, excessive current in the stitching vias can lead to via failures losing connection. Any number of power supply nets with source and sink models can be simulated together.

You can use the default empty template available in the project panel. Expand the **DC Analysis 1** option present in the **Analyses** node:

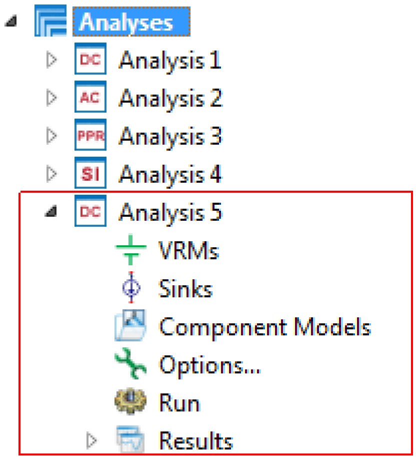


You can also create a new analysis. To create a new analysis:

1. Right-click **Analyses** and select **New PI-DC Analysis**.



2. Expand the analysis tree node of the new analysis branch:

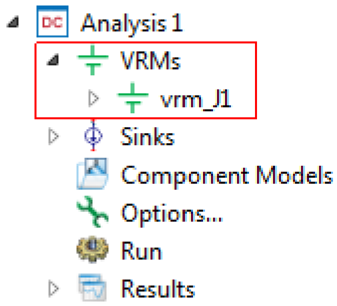


Defining a VRM

A VRM regulates the voltage provided to a microprocessor.

To define a VRM:

1. Select a VRM (or connector) component instance in the Parts tree. You can also search the required component. For more information about, see How to Find a Component Instance.
2. Drag and drop the selected instance on the **VRMs** part in the project panel. A new VRM definition is added in the project panel.



3. If the VRM connects to multiple power nets, the **Select Net for Instance** dialog box is displayed, where you have to select the required net. The same applies for ground nets.
4. Double click the VRM definition to open the VRM Editor dialog box.
5. Specify the **DC Voltage**, **Voltage Tolerance**, and **Resistance**. The **Inductance** will not play a role for a DC analysis. The Voltage Tolerance (absolute or in percent) is the voltage drop that is allowed at the output pins of the VRM. There will be no voltage drop in the VRM in case the series resistance and inductance values are 0.
6. Click **Done**.

Defining a Sink

A sink is a component that reduces the temperature of an electronic device by dissipating heat into the surrounding air. You can define the following options for a sink:

To define a sink:

1. Find the IC component instance(s) in the Components list.
2. Select the instance(s) and drag and drop them onto **Sinks**. New Sink definitions will be added in the tree.

3. In case the IC connects to multiple power nets, the **Select Net for Instance** dialog box is displayed, where you can select the required net.
4. In case multiple instances were dropped, the **Select Net for Multiple Instances** dialog box provides a check box to **Use same Selection for All Sinks/VRMs**. The same applies for ground nets.
5. Double-click the Sink definition to open the Sink Editor dialog box.
6. Specify the **DC Current**, **Resistance** and **Voltage Tolerance**. The Voltage Tolerance (absolute or in percent) is maximum voltage drop that is allowed at the input pins of the IC.

Defining Options

A PI-DC analysis provides following options:

Option	Description
Use Ideal Ground Approximation	When enabled, this option will treat all Ground nets as ideal shorts. The power rail is often the dominant contributor to the voltage drop. This option provides you the capability to analyze this dominant factor in less simulation time.

Running the PI-DC Analysis

After saving your analysis setup, double click **Run** to start the PI-DC analysis. The Simulations window is displayed and the PI-DC analysis will start. The Simulations window allows you to monitor and manage simulations.

Show Me How Do I Perform a PI-DC Analysis

Video: How to Perform a PI-DC Analysis

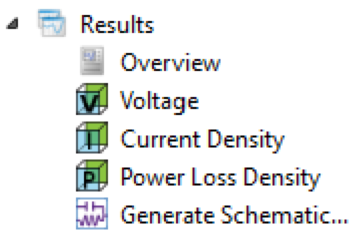
Viewing PI-DC Analysis Results

After completing a PI-DC analysis setup, you can run the analysis in SIPro/PIPro. You can view the overview, voltage, current density, and power loss density. You can also generate a schematic.

Running a PI-DC Analysis

To run a PI-DC analysis and view results:

1. Save your PI-DC analysis setup.
2. Double-click **Run** to start the PI-DC analysis. The Simulations window is displayed, which allows you to monitor and manage simulations. The results are listed in the project panel.
3. Expand **Results** in the Analysis list of the project panel:



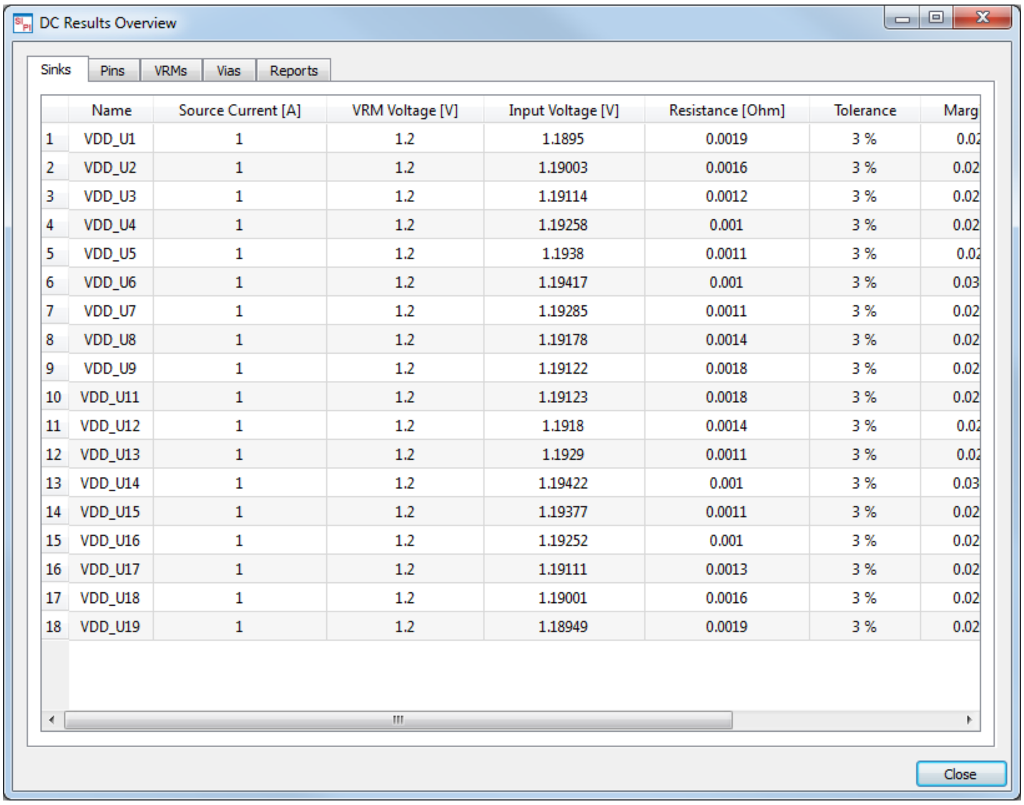
4. Double-click the required analysis result.

Overview

Clicking the **Overview** option in the Results list opens the DC Results Overview window. This window provides a table-based overview of voltages and currents at Sinks, Pins, VRMs, and Vias.

Sinks

In the Sinks tab, you can view the source current, VRM voltage, input voltage, resistance, tolerance, and margin values.

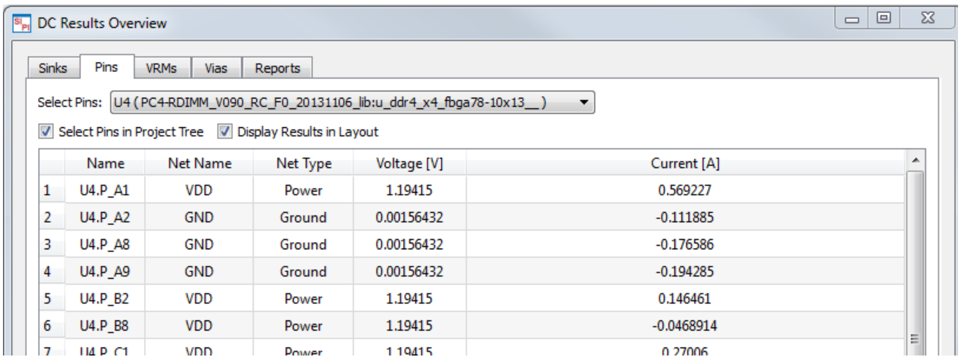


	Name	Source Current [A]	VRM Voltage [V]	Input Voltage [V]	Resistance [Ohm]	Tolerance	Marg
1	VDD_U1	1	1.2	1.1895	0.0019	3 %	0.0
2	VDD_U2	1	1.2	1.19003	0.0016	3 %	0.02
3	VDD_U3	1	1.2	1.19114	0.0012	3 %	0.02
4	VDD_U4	1	1.2	1.19258	0.001	3 %	0.02
5	VDD_U5	1	1.2	1.1938	0.0011	3 %	0.0
6	VDD_U6	1	1.2	1.19417	0.001	3 %	0.03
7	VDD_U7	1	1.2	1.19285	0.0011	3 %	0.02
8	VDD_U8	1	1.2	1.19178	0.0014	3 %	0.02
9	VDD_U9	1	1.2	1.19122	0.0018	3 %	0.02
10	VDD_U11	1	1.2	1.19123	0.0018	3 %	0.02
11	VDD_U12	1	1.2	1.1918	0.0014	3 %	0.0
12	VDD_U13	1	1.2	1.1929	0.0011	3 %	0.0
13	VDD_U14	1	1.2	1.19422	0.001	3 %	0.03
14	VDD_U15	1	1.2	1.19377	0.0011	3 %	0.02
15	VDD_U16	1	1.2	1.19252	0.001	3 %	0.02
16	VDD_U17	1	1.2	1.19111	0.0013	3 %	0.02
17	VDD_U18	1	1.2	1.19001	0.0016	3 %	0.02
18	VDD_U19	1	1.2	1.18949	0.0019	3 %	0.02

Pins

To view pins:

1. Click the **Pins** tab in the DC Results Overview window.
2. Choose the required pin from the **Select Pins** drop-down list. The table is updated with the selected pin results.



VRMs

To view VRMs, click the **VRMs** tab in the DC Results Overview window.

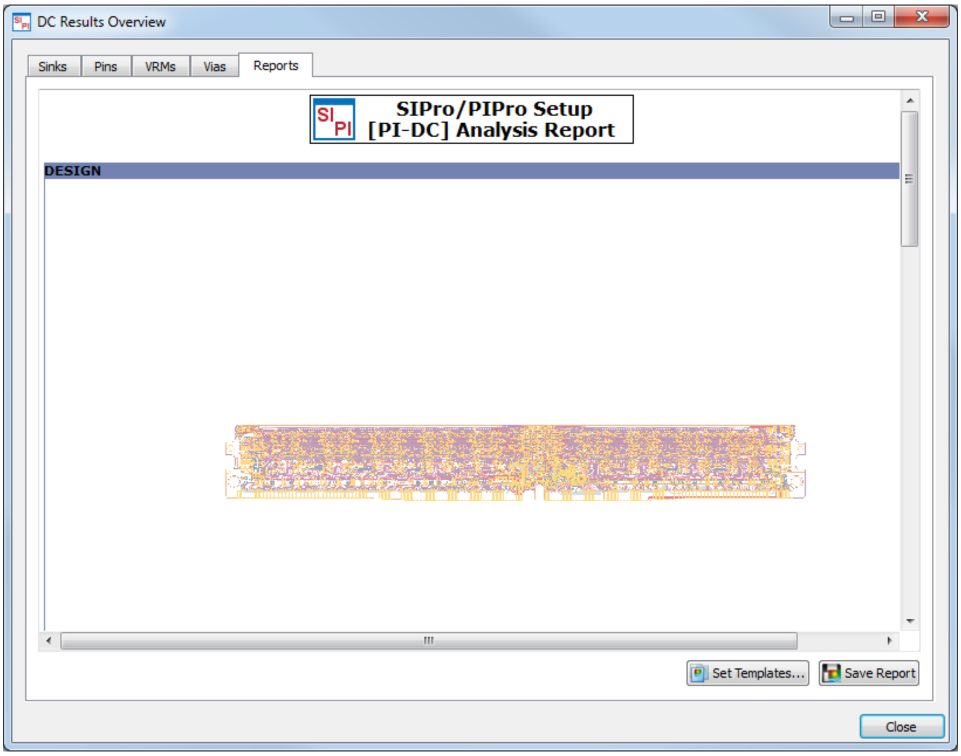
Vias

To view ID and current of vias, click the **Vias** tab.

Reports

To generate a PI-DC analysis report:

1. Click the **Reports** tab.



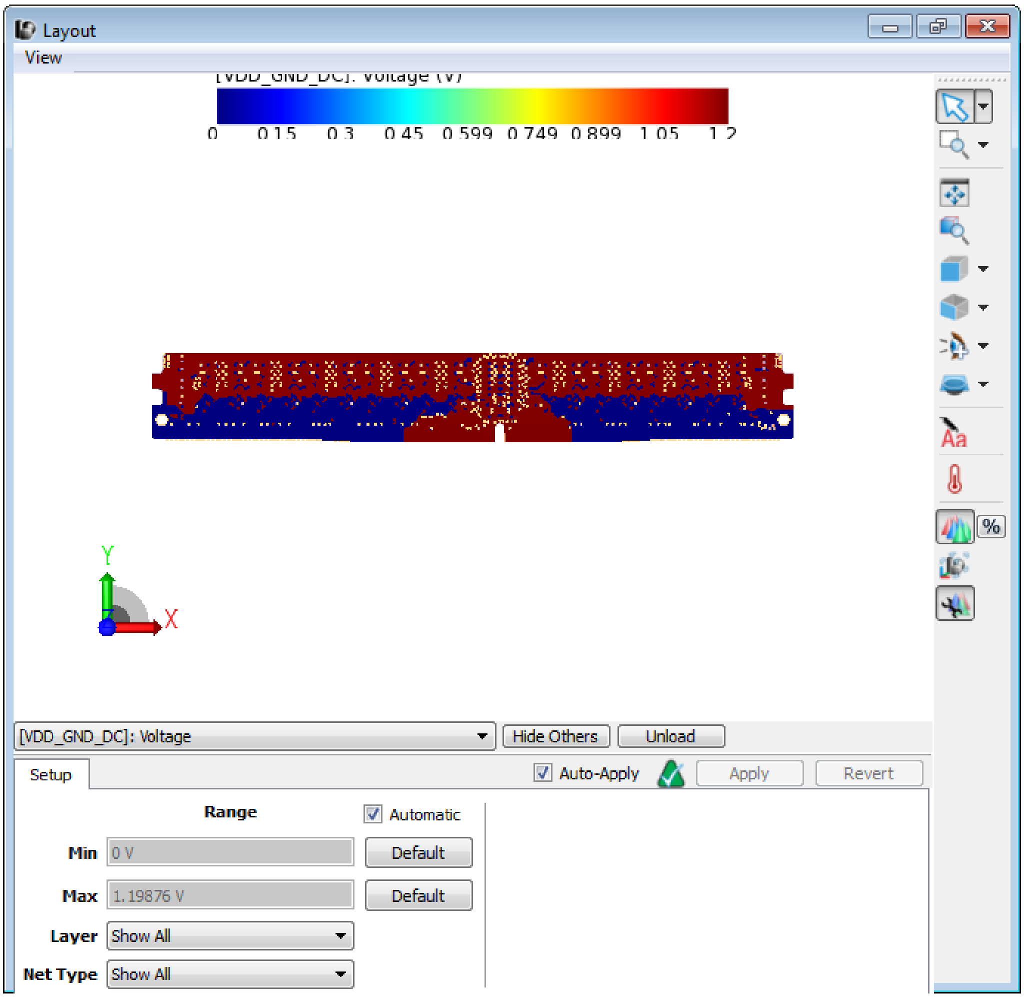
2. Click **Save Report**.

Setting Template

To set a template for your report, click the **Set Templates** button. Browse to the DocX, ODT, and HTML location for selecting a template for your report.

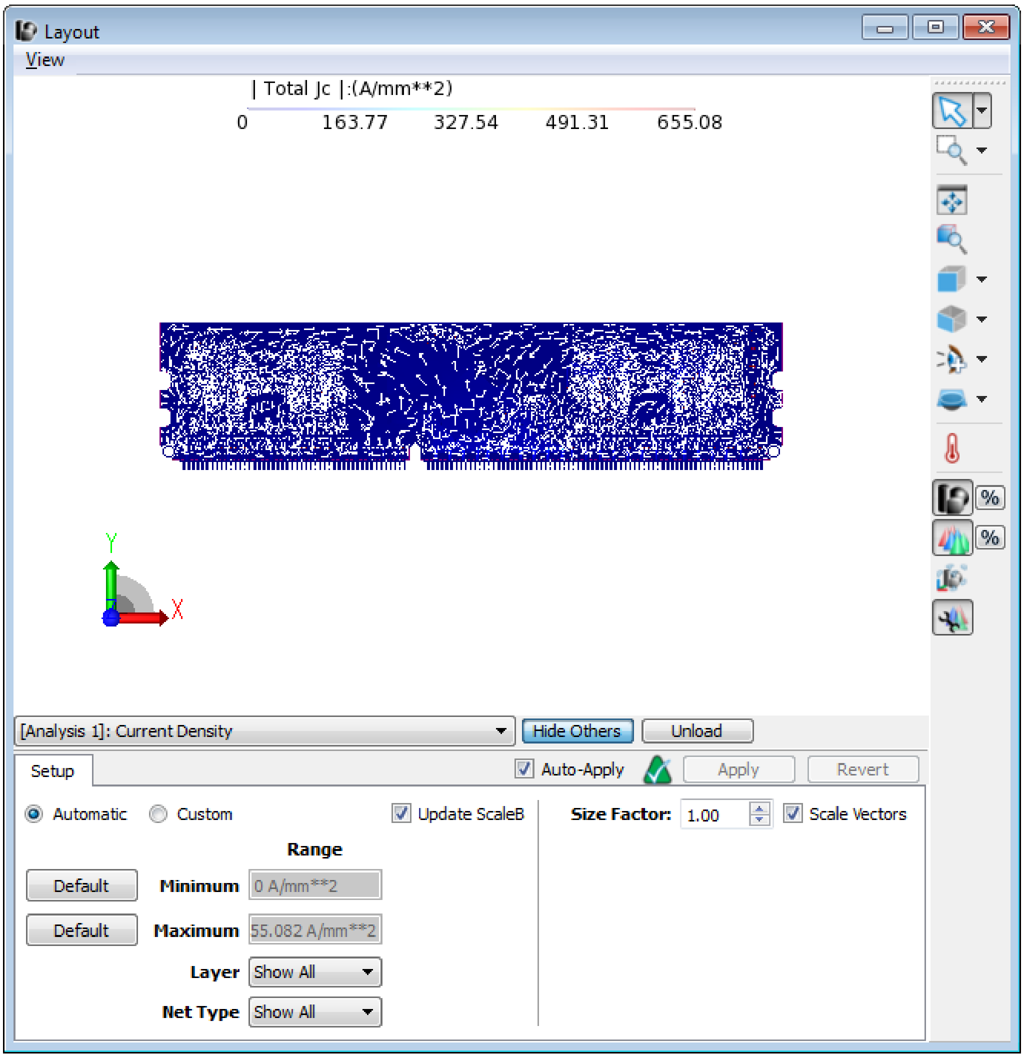
Voltage

Clicking the **Voltage** option in the Results list loads the voltage plot. This is a scalar plot. The voltage is forced at 0 at the negative pins of the VRMs.



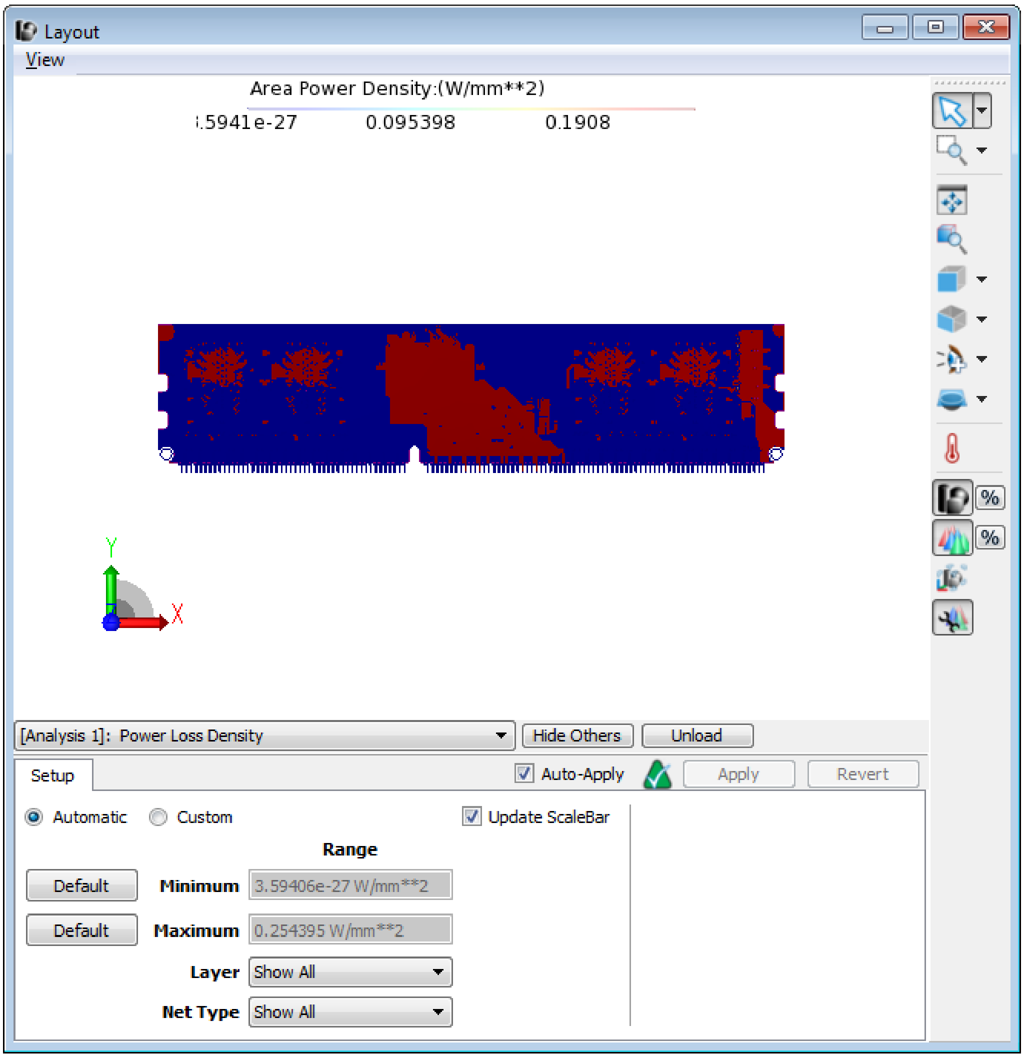
Current Density

Clicking the **Current Density** option in the Results list loads the current density plot. This is a vector plot. The color indicates the current density amplitude. The white arrow indicates the direction of the current. The arrow size is not an indication for the current density amplitude.



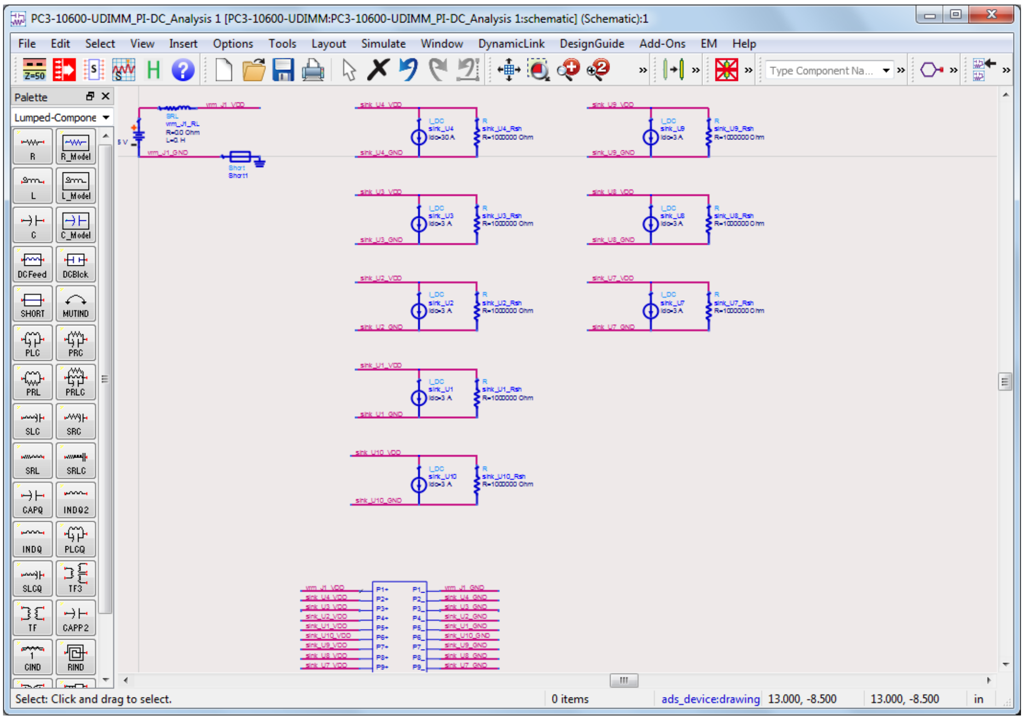
Power Loss Density

Clicking the **Power Loss Density** option in the Results list loads the power loss density. This is a scalar plot. This plot shows the power loss per unit area.



Generate Schematic

Clicking the **Generate Schematic** option opens a Schematic window with the VRMs, Sinks, Component Models and an SNP that points to the DC S-parameter model.



See Also

- Setup Options for Customizing Results

PI-AC Dynamic IR Drop Analysis

PI-AC Dynamic IR Drop Analysis

- Creating a PI-AC Analysis Setup
- Viewing PI-AC Analysis Results

Creating a PI-AC Analysis Setup

A PI-AC analysis computes the impedance for the IC current loads over a broad frequency range. It helps you to identify whether the power distribution network (PDN) provides a low impedance path from the Voltage Regulator Module (VRM) to the IC. An excessive impedance in a certain frequency range can generate excessive voltage noise, also called dynamic IR drop, when the IC power supply pins draw large amounts of transient current, required for I/O or core logic switching, at rates that fall into that frequency range.

Defining a PI-AC Analysis Setup

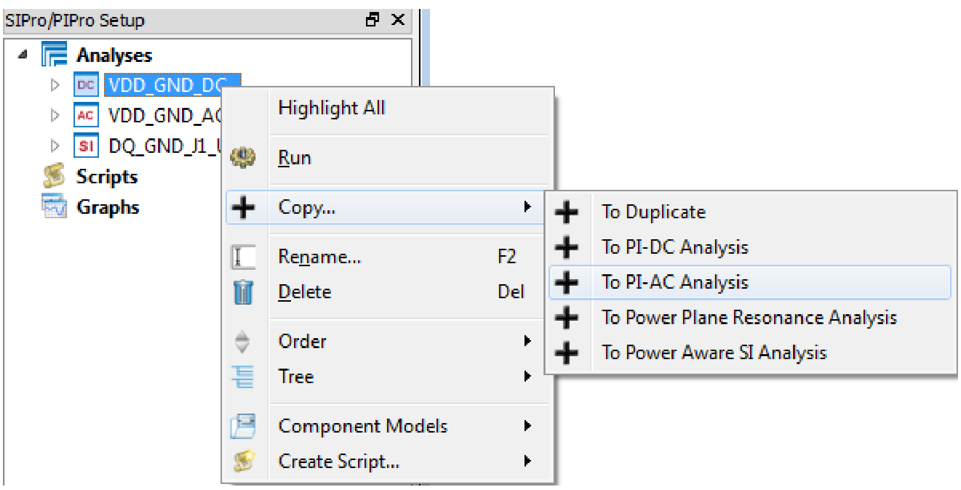
You can create a PI-AC analysis setup in the following ways:

- Copying an existing PI-DC analysis setup.
- Creating a new PI-AC analysis setup.

Copying the PI-DC Analysis Setup

If you have already performed a PI-DC analysis, you can copy its setup for creating a PI-AC analysis setup. To copy an existing setup:

1. Right-click the PI-DC analysis setup in the project panel.
2. Select **Copy** > **To PI-AC Analysis**.



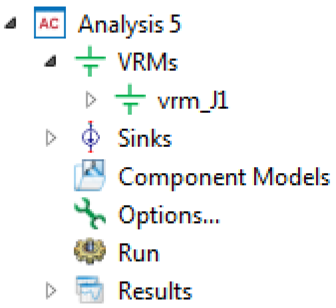
Creating a New PI-AC Analysis Setup

If you want to create a new setup, right-click **Analyses** and then select **New PI-AC Analysis**. After creating a new setup, you need to define VRMs and sinks. For more information, see PIPro Analysis Setup Overview.

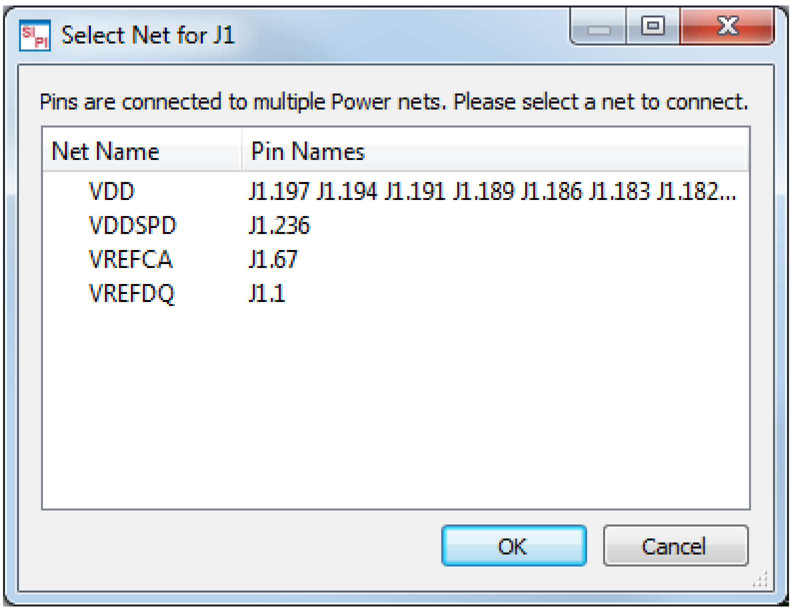
Defining VRMs

To define a VRM:

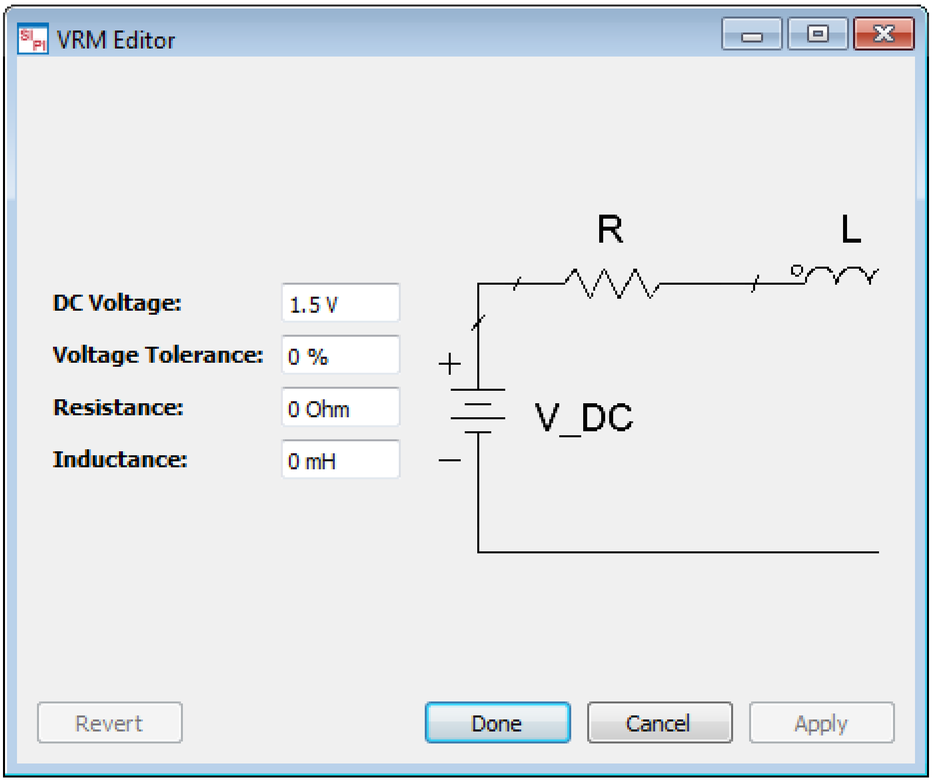
1. Select a VRM (or connector) component instance in the **Components** list. You can also search the required component. For more information about, see How to Find a Component Instance.
2. Drag and drop the selected instance on **VRMs** in the AC Analysis list. A new VRM definition is added.



3. If the VRM connects to multiple power nets, the Select Net for Instance dialog box is displayed. Select a net. Similarly, you can select ground nets.



4. Double-click the VRM definition to open the VRM Editor dialog box.



5. Specify the **DC Voltage**, **Voltage Tolerance**, and **Resistance**. The **Inductance** will not play a role for a DC analysis. The Voltage Tolerance (absolute or in percent) is the voltage drop that is allowed at the output pins of the VRM. There will be no voltage drop in the VRM in case the series resistance and inductance values are 0.

Defining Sinks

To define a sink:

1. Find the IC component instance(s) in the **Components** list.
2. Select the instance(s) and drag and drop them onto **Sinks**. New Sink definitions will be added in the tree.
3. In case the IC connects to multiple power nets, the Select Net for Instance dialog pops up where you will have to select one.
4. In case multiple instances were dropped, the Select Net for Multiple Instances dialog provides a check box to **Use same Selection for All Sinks/VRMs**. The same applies for ground nets.
5. Double click the Sink definition to open the Sink Editor dialog. Specify the **DC Current**, **Resistance** and **Voltage Tolerance**. The Voltage Tolerance (absolute or in percent) is maximum voltage drop that is allowed at the input pins of the IC.

Defining Component Models

For a PI-AC analysis, you need to consider the decoupling capacitors because they are not an open circuit at AC.

To define a component model:

1. Select the component group, which you want to analyze, in the **Components** list.
2. Drag and drop the component group on **Component Models**. The instances will be grouped by component name.
3. Define the electrical model parameters by using one of the following ways:

-
    - Double-click the component instance to display the Component Model Editor window. For more information, see Creating and Editing Component Models for Analysis.
    - Alternatively, you can open the Vendor Parts database by selecting **Tools > Vendor Parts DB**. Drag and drop the appropriate vendor part onto the component instance node (single model assignment) or onto the component model group node (group model assignment). For more information, see Using Vendor Parts DB Browser.

Defining Options

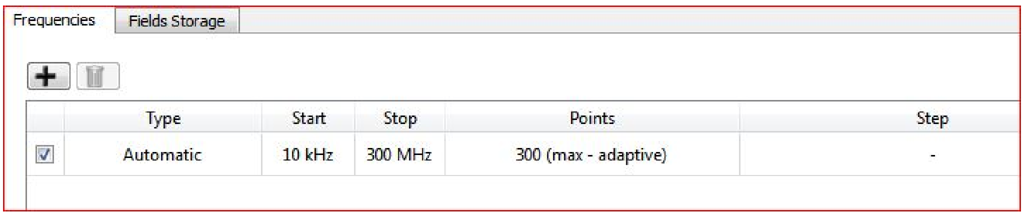
To define options, double-click **Options** in the PI-AC Analysis list in the project panel. provides It provides the following options:

Option	**Description**
Resolution	Points closer than the specified resolution will be considered equal during the simulation
Arc Resolution	Value used to discretize circles and arcs during the simulation
Custom Target Mesh Size	Mesh size used to generate the global 3D mesh during simulation
Use Ideal Ground Approximation	When enabled, this option specifies that all Ground nets are considered as ideal shorts. The power rail is often the dominant contributor to the voltage drop. This option provides you the capability to analyze this dominant factor in less simulation time.

Defining Frequency Plans

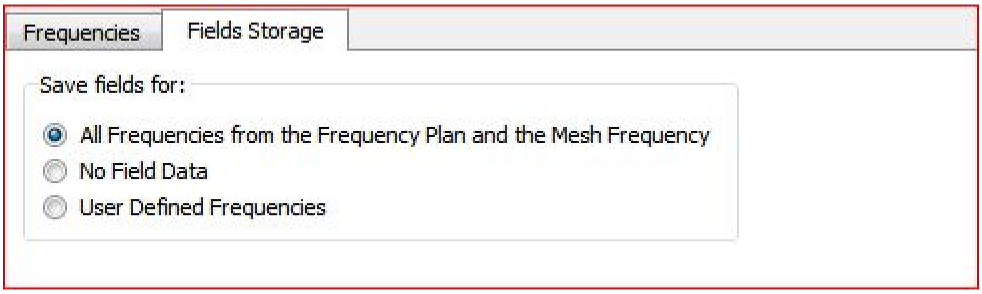
By default, you can use the Automatic, Logarithmic and Adaptive frequency plan. The Automatic will provide the optimal frequency sampling using a combination of Adaptive and Logarithmic sweeps.

Click the **Frequency Plans** tab to specify a plan.



Saving fields for field plots

By default, the fields valued needed for visualization are not stored. You can enable this using the Field Storage dialog.



Note that this will save big files to disc. Depending on the example this can result in multiple GB of saved data.

Saving the Setup

Save your PI-AC analysis setup and double-click **Run** to start the PI-AC analysis.

Show Me How Do I Perform a PI-AC Analysis

Video: How to Perform a PI-AC Analysis

Viewing PI-AC Analysis Results

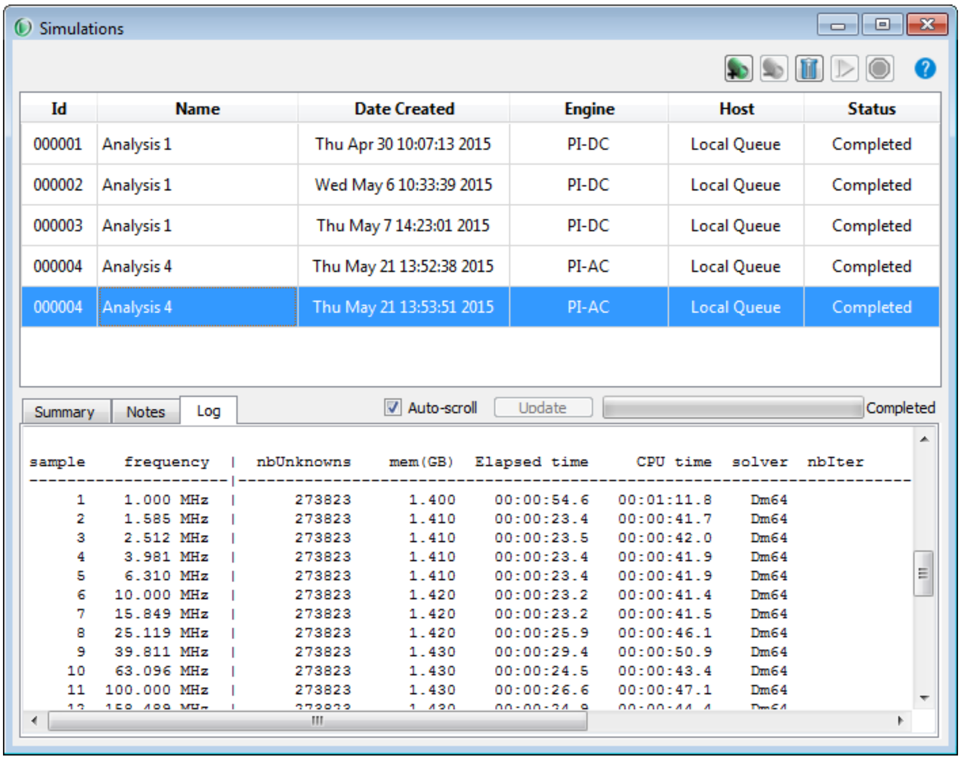
After creating a PI-AC analysis setup, you can run the analysis in the SIPro/PIPro Setup window. For a PI-AC analysis, the following results are displayed:

- PDN Impedance
- S-parameters
- Electric Field
- Magnetic Field
- Current Density
- Generate Test Bench
- Generate Sub Circuit

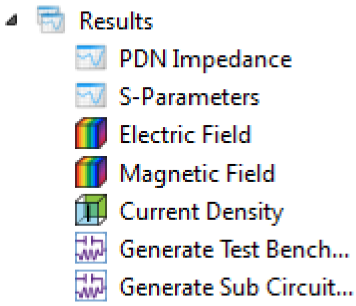
Running a PI-AC Analysis

To run a PI-AC analysis and view results:

1. Save your PI-AC analysis setup.
2. Double-click **Run** to start the PI-AC analysis. The Simulations window is displayed, which allows you to monitor and manage simulations:



3. Expand the **Results** tree item.
4. Double-click the required result:



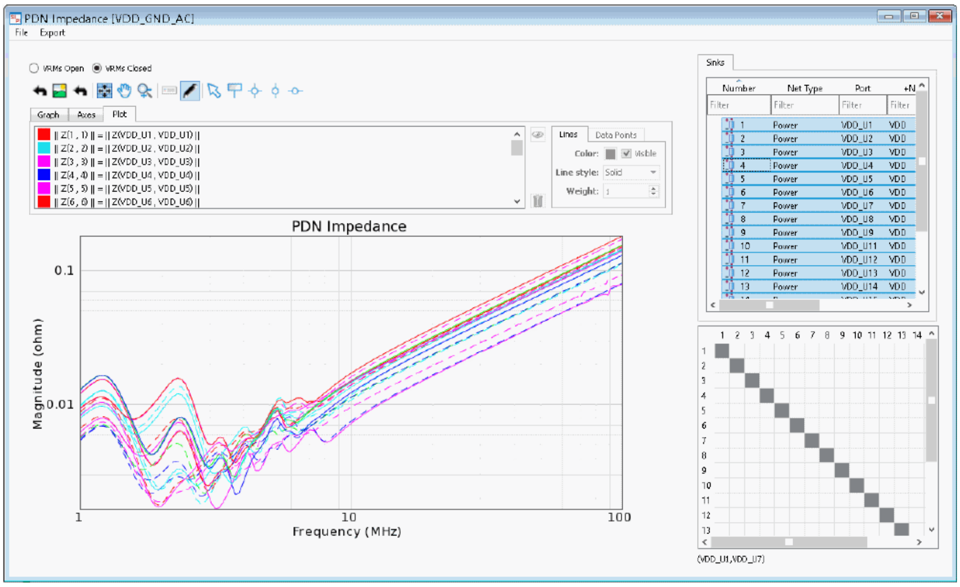
PDN Impedance

A PI-AC analysis computes the impedance of the IC current loads over a broad frequency range. You can identify whether the power distribution network (PDN) provides a low impedance path from the Voltage Regulator Module (VRM) to the sinks or ICs. It provides the input impedance value from each sink.

Viewing PDN Impedance

To view PDN impedance results:

1. In the **Results** list, double-click **PDN Impedance**. The PDN impedance window is displayed.
2. Select the required sinks.
3. Right-click the selected sinks and choose **Add PDN Impedance**.



Removing PDN Impedance Plots

To remove PDN Impedance plots:

1. Click the **Plot** tab.
2. Select the plots to be removed.
3. Click  . The selected plots are deleted from the PDN Impedance window.

Importing a Target Impedance File

The PDN impedance window supports the import and visualization of target impedances. The following table lists the contents of a sample CSV file:

Frequency[Hz]	**Impedance[Ohm]**
100000	0.01
1000000	0.01
1000000	0.1
10000000	0.1
10000000	1.5
1E+08	1.5
1E+08	2
1E+09	2

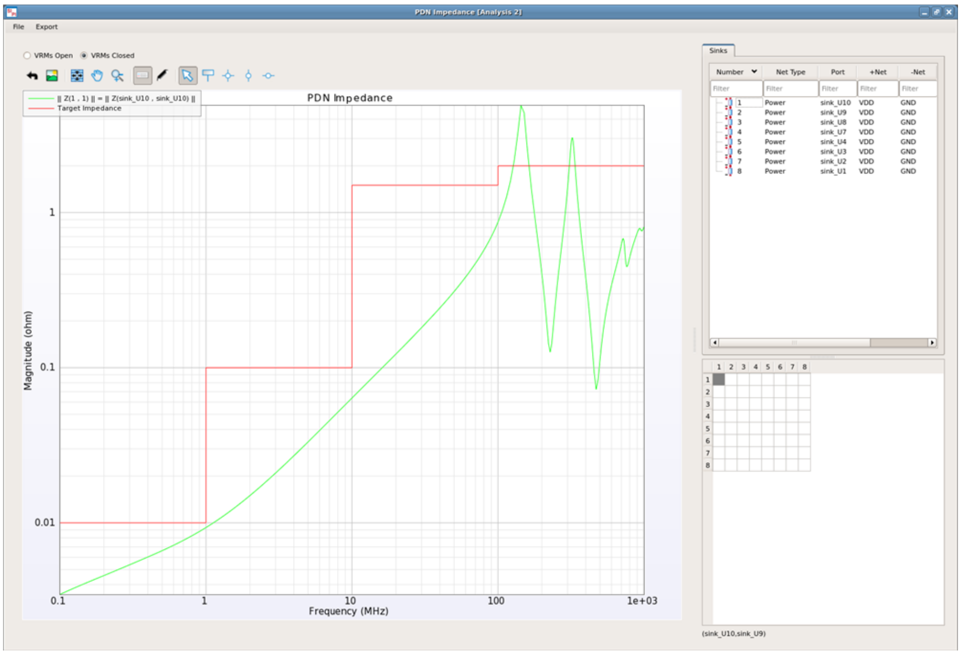
Frequency[Hz]	Impedance[Ohm]
1E+09	2

To import a series of target impedances:

1. Select **File** > **Import Target Impedance**. The Import Custom Target Impedance File dialog box is displayed.
2. Select a **CSV** file containing values. This file should contain comma separated frequency and impedance values.

   NOTE The importer does not perform a validity check on the actual data.

3. Once imported, the values are added as a new plot on the impedance graph, as shown in the following figure:



Result Type Options

In the Result Type section, you can select the following options:

- **Impedance**: Displays the PDN impedance values.
- **S-Parameters**: Displays the s parameter results.

The impedance plots show the PDN impedance values including the default component models specified in the setup. After changing the default model specified in the Component Model Editor dialog (see Creating and Editing Component Models for Analysis), you can update the impedance plots without recalculation. This allows a quick way to vary e.g. decap component models and inspect the effect on the PDN plots.

VRM Options

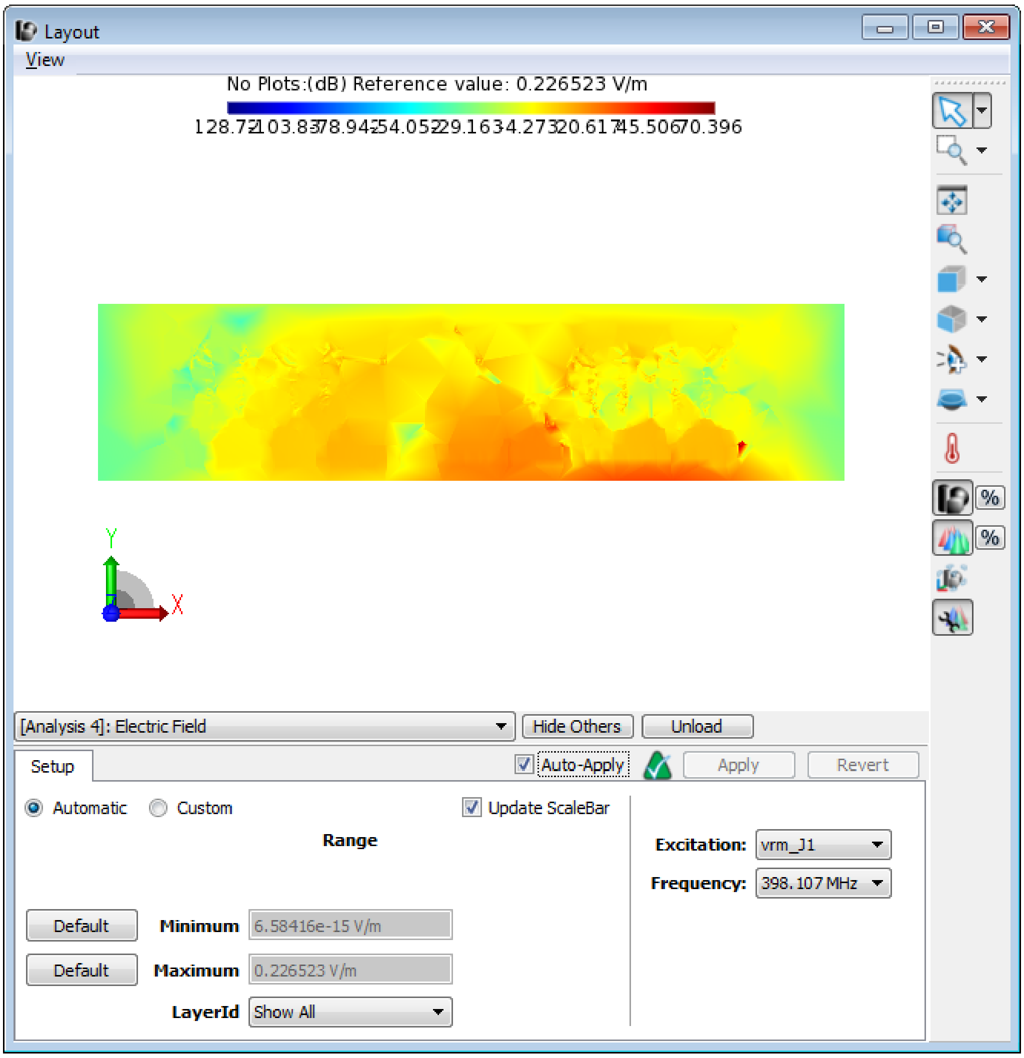
If you are viewing Impedance results, you can also select the following VRMs options:

- **Open**

- **Closed**

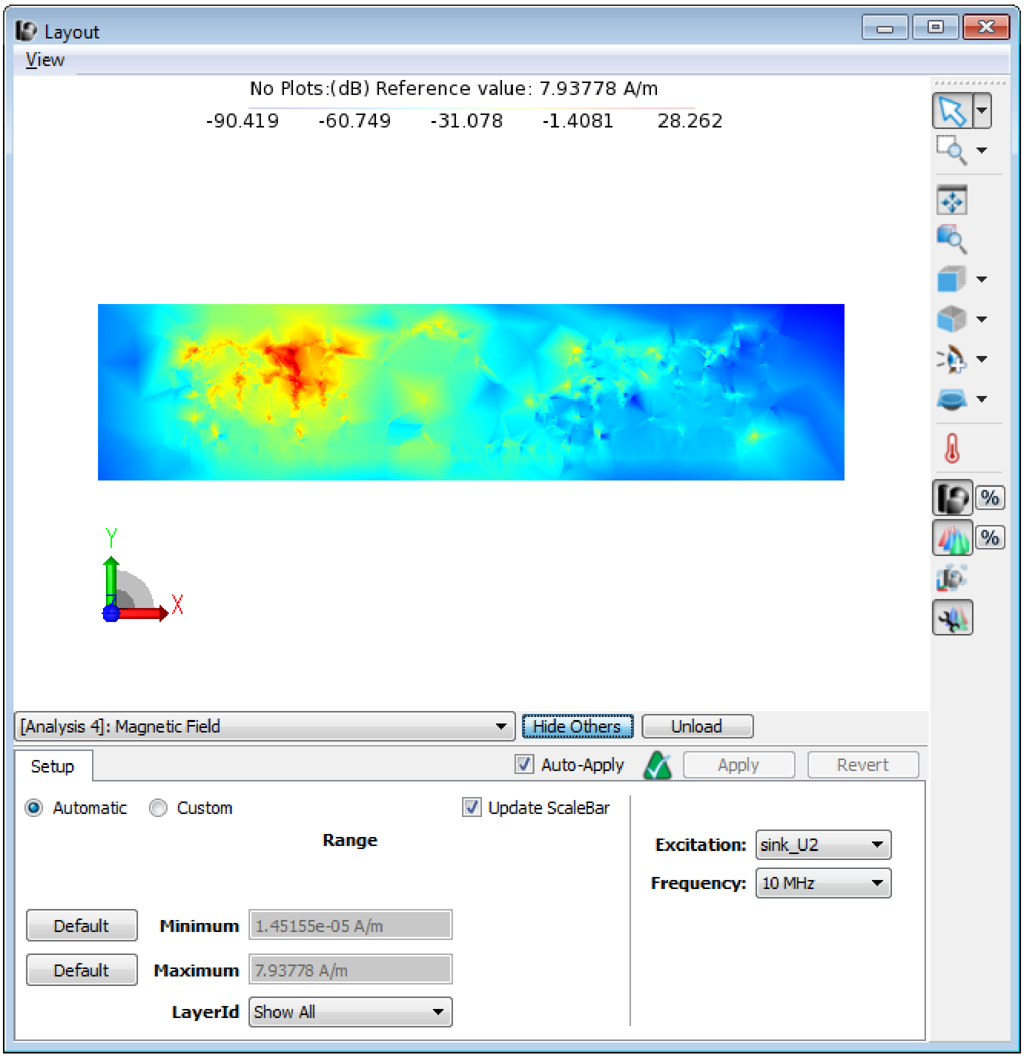
Viewing Electric Field

Double-click **Electric Field** in the **Results** list of PI AC analysis. The electric field plot is displayed:



Viewing Magnetic Field

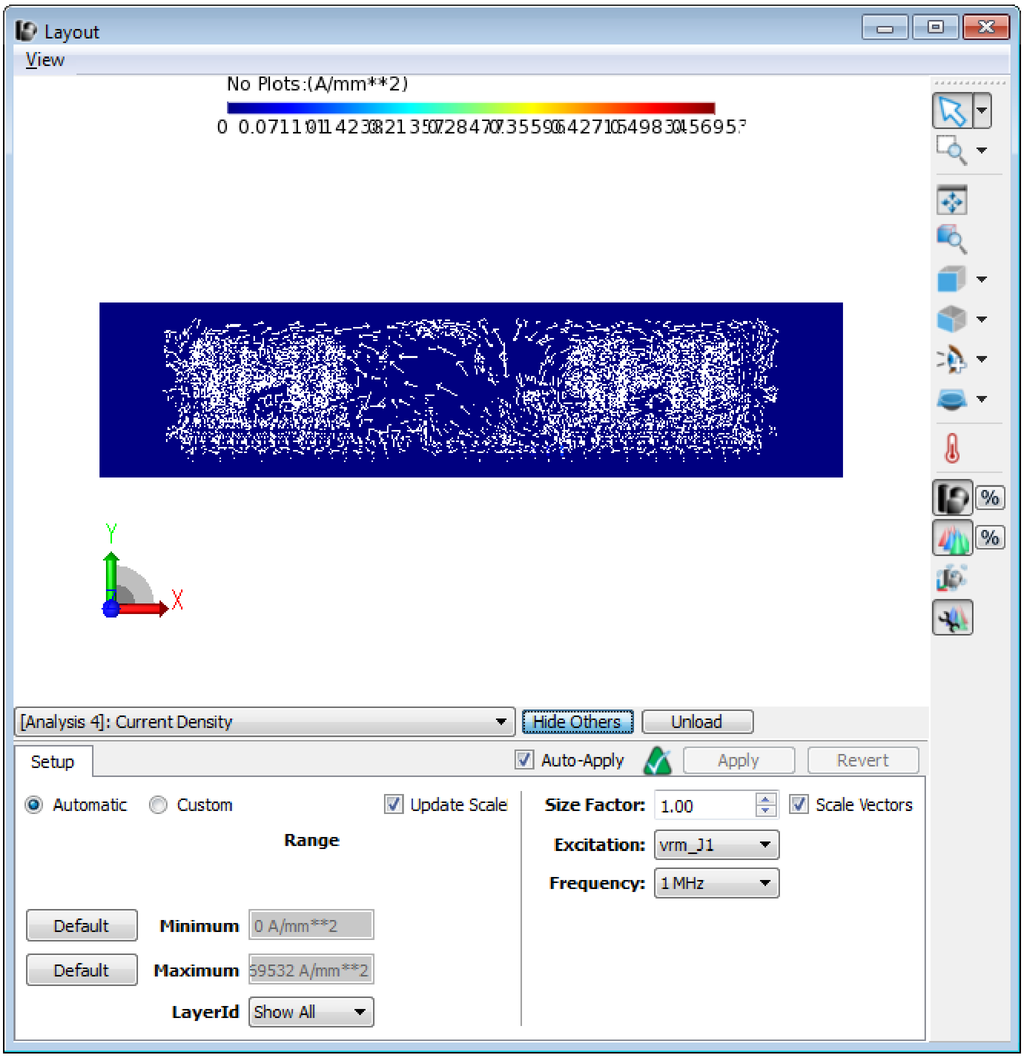
Double-click **Magnetic Field** in the Results list of PI AC analysis. The magnetic field plot is displayed:



NOTE If you want to view only the recent results and hide previous analysis, click **Hide Others**.

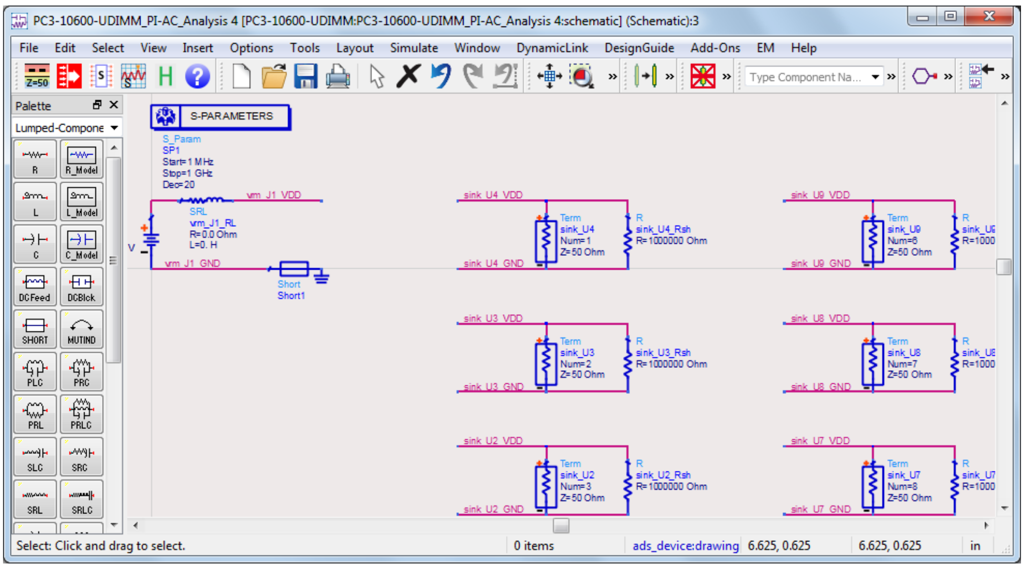
Current Density

Double-click **Current Density** in the Results list of PI AC analysis.



Generating a Schematic

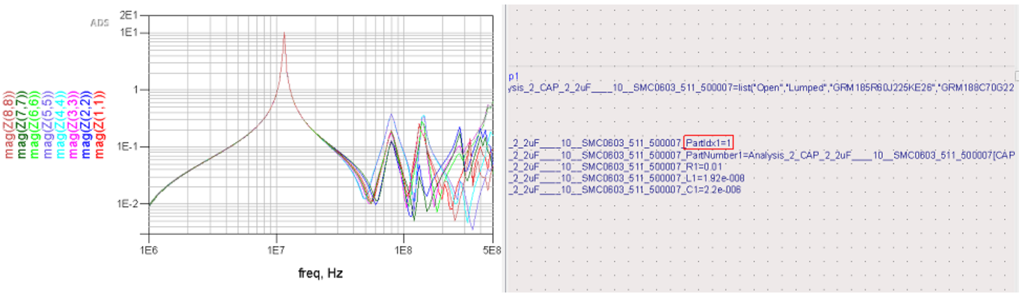
Double-click **Generate Schematic** in the Results list of PI-AC analysis. A Schematic window is displayed, which includes the VRMs, Sinks, Component Models and an SNP that points to the S-parameter model.



You can run a simulation from the schematic. The results display the input impedance for sinks, which is similar to the SIPro/PIPro results.

Enable the decoupling capacitors

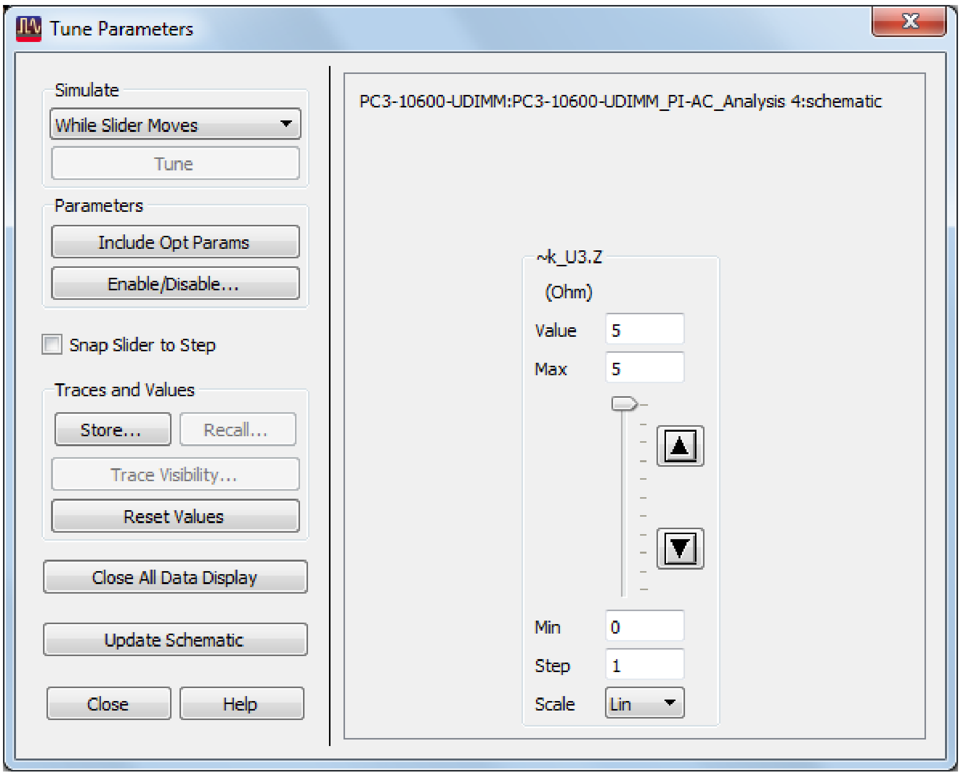
The key parameter is the index, which determines the location of the switch. If the index is 0, the switch is open. If the value of index is set to 1, the connection is made to the first part. Similarly, if the index is 2, then connection is made to the second part. The following figure displays the results when the index value is set to 1:



Tuning Parameters

To tune parameters:

1. Click the**Tuning** icon. The **TuneParameters** dialog box is displayed.
2. Select the decoupling capacitor in Schematic.
3. Set the Minimum value to **0**.
4. Set the maximum value to **5**.
5. Specify the Step value as **1**.



6. Click the up and down arrows.

See Also

- Setup Options for Customizing Results

PI-PPR Analysis

PI-PPR Analysis

- Creating a PI-PPR Analysis Setup
- Viewing PI-PPR Analysis Results

Creating a PI-PPR Analysis Setup

A PI-PPR (Power Plane Resonance) analysis computes the resonances of the design over a broad frequency range. It helps you to identify the resonance frequencies and fields of the design, with and without components. It provides a global view on natural modes (eigenmodes) between the power and ground planes. Some of the modes may cause excessive impedance from VRM to IC. Some modes resonating around the signal frequencies may affect the signal quality.

Defining a PI-PPR Analysis Setup

You can create a PI-PPR analysis setup in the following ways:

- Copying an existingPI-DC/AC analysis setup.
- Creating a new PI-PPR analysis setup.

Copying the PI DC/AC Analysis Setup

If you have already performed a PI-DC or PI-AC analysis, you can copy its setup for creating a PI-PPR analysis setup. To copy an existing setup:

1. Right-click the existing **PI-DC** or **PI-AC** analysis setup in the project panel.
2. Select **Copy** > **To PI-PPR Analysis**. The entire setup is copied to create a new PI-PPR analysis setup.

Creating a New PI-PPR Analysis Setup

If you want to create a new setup, right-click **Analyses** and then select **New PI-PPR Analysis**.

NOTE Definition of VRMs and sinks are optional in PI-PPR. At least one VRM, or one sink or one component is suggested to be defined to indicate the power and ground nets be put in PPR simulation. For more information, see PIPro Analysis Setup Overview.

Defining VRMs

To define a VRM:

1. Select a VRM (or connector) component instance in the Parts tree. You can also search the required component. For more information about, see How to Find a Component Instance.
2. Drag and drop the selected instance on the VRMs part in the project panel. A new VRM definition is added in the project panel.

3. If the VRM connects to multiple power nets, the Select Net for Instance dialog pops up where you will have to select one. The same applies for ground nets.
4. Double-click the VRM definition to open the VRM Editor dialog box.
5. Specify the DC Voltage, Voltage Tolerance, and Resistance. The Inductance will not play a role for a DC analysis. The Voltage Tolerance (absolute or in percent) is the voltage drop that is allowed at the output pins of the VRM. There will be no voltage drop in the VRM in case the series resistance and inductance values are 0.

Defining Sinks

To define a sink:

1. Find the IC component instance(s) in the Parts tree.
2. Select the instance(s) and drag and drop them onto Sinks. New Sink definitions will be added in the tree.
3. In case the IC connects to multiple power nets, the Select Net for Instance dialog pops up where you will have to select one.
4. In case multiple instances were dropped, the Select Net for Multiple Instances dialog provides a check box to Use same Selection for All Sinks/VRMs. The same applies for ground nets.
5. Double click the Sink definition to open the Sink Editor dialog. Specify the DC Current, Resistance and Voltage Tolerance. The Voltage Tolerance (absolute or in percent) is maximum voltage drop that is allowed at the input pins of the IC.

Defining Component Models

For a PI-**PPR** analysis, you need to consider the decoupling capacitors because the R, L and C parameters associated with that component may affect the resonances.

To define a component model:

1. Select the component group, which you want to analyze, in the **Components** list.
2. Drag and drop the component group on **Component Models**. The instances will be grouped by component name.
3. Define the electrical model parameters by using one of the following ways:
    a. Double-click the component instance to specify the electrical model parameters for that instance.
    b. Alternatively, you can open the Vendor Parts database by selecting Tools > Vendor Parts DB. Drag and drop the appropriate vendor part onto the component instance node (single model assignment) or onto the component model group node (group model assignment). For more information, see Using Vendor Parts DB Browser.

Defining Options

The same options can be set as for an PI-AC simulation.

Defining Frequency Plan

There are three parameters in PPR frequency plan: **Start frequency (Fstart)**, **Stop frequency (Fstop)**, and **Number of eigenmodes (N)**. The first N modes with resonant frequencies between Fstart and Fstop would be provided. By default, the Start and Stop frequencies are defined as minFreq and maxFreq. You can change the minFreq, maxFreq, by using the Parameters window. For more information, see Parameters window.

Saving the Setup

Save your PI-PPR analysis setup and double-click **Run** to start the PI-PPR analysis.

Viewing PI-PPR Analysis Results

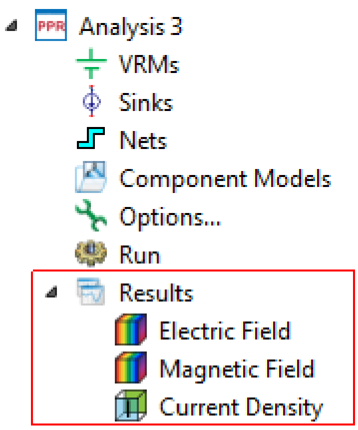
After creating a PI-PPR analysis setup, you can run the analysis in the SIPro/PIPro Setup window. For a PI-PPR analysis, the following results are displayed:

- Electric Field
- Magnetic Field
- Current Density

Running a PI-PPR Analysis

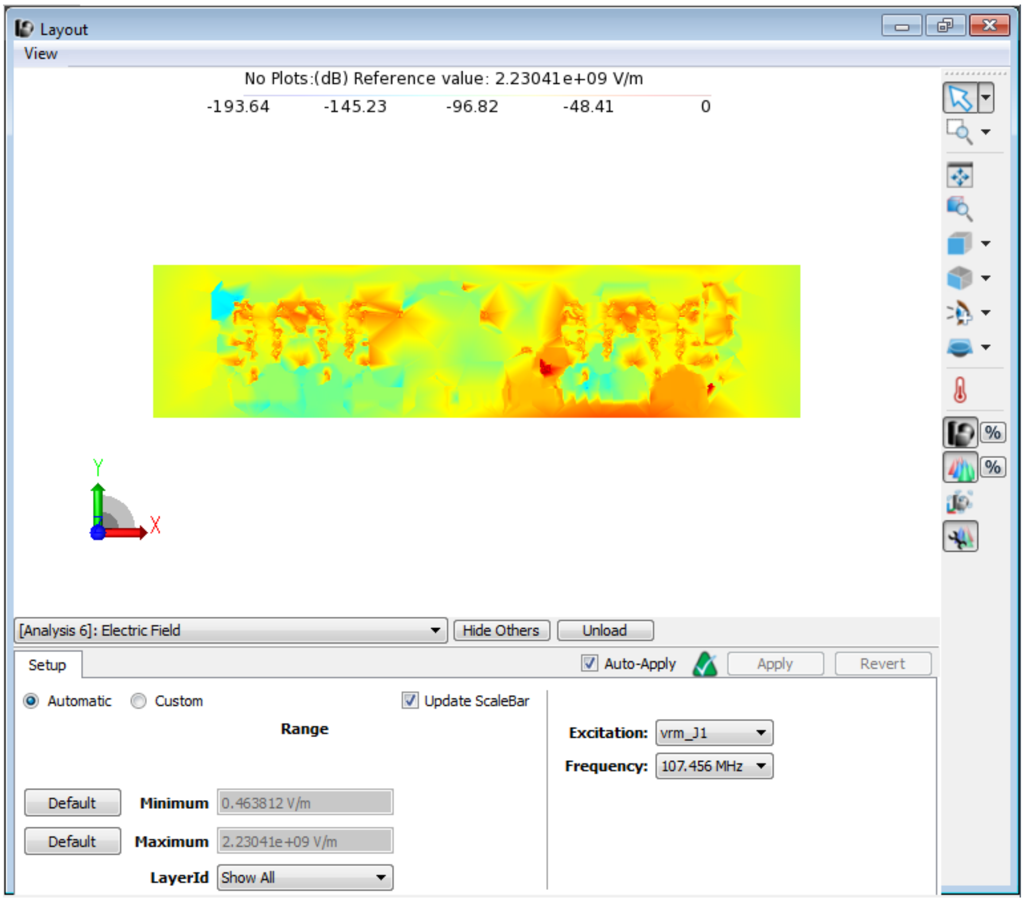
To run a PI-PPR analysis and view results:

1. Save your PI-PPR analysis setup.
2. Double-click **Run** to start the PI-PPR analysis. The Simulations window is displayed, which allows you to monitor and manage simulations.
3. Expand the **Results** tree item.
4. Double-click the required result:



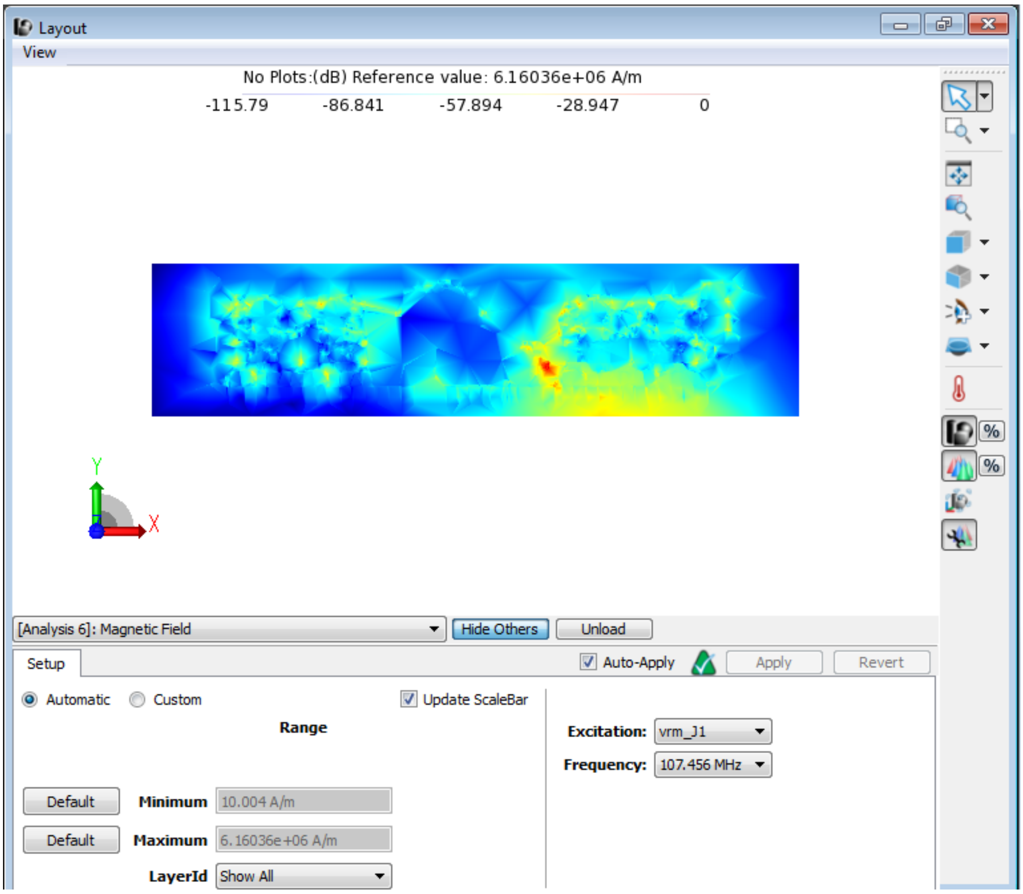
Electric Field

Double-click **Electric Field** in the **Results** list of PI-PPR analysis. The electric field plot is displayed:



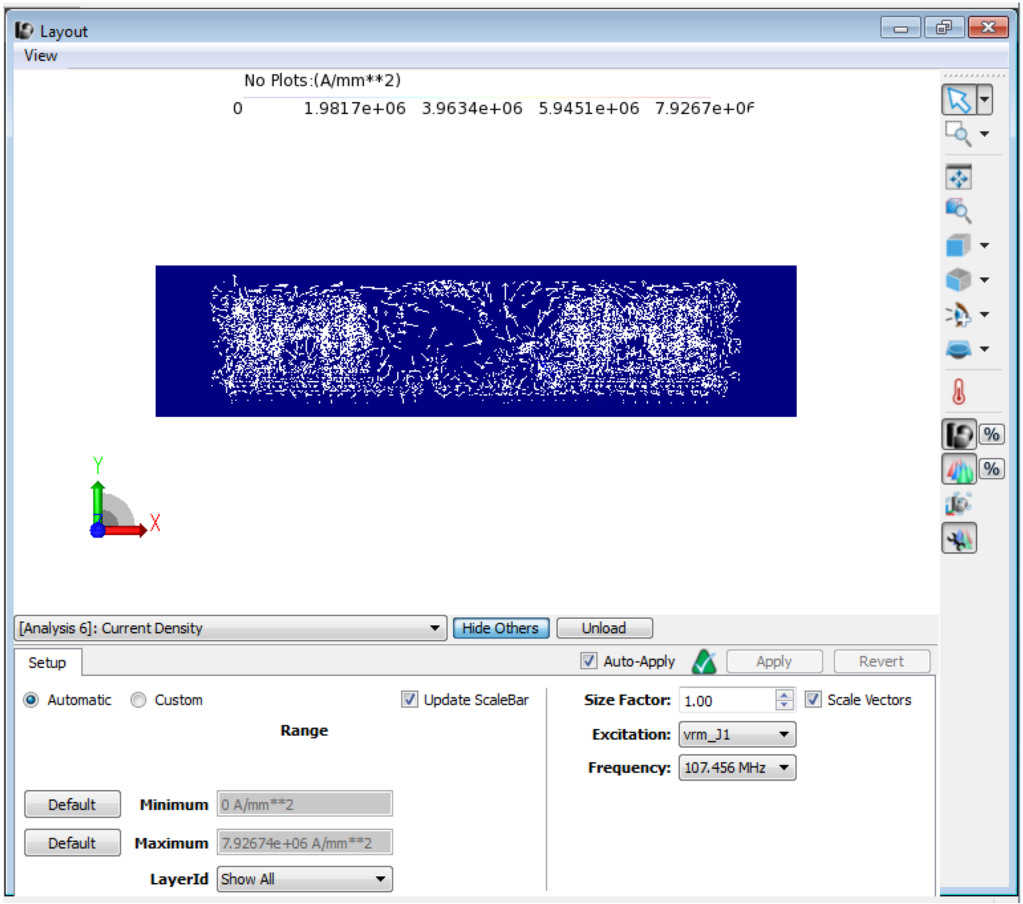
Magnetic Field

Double-click **Magnetic Field** in the Results list of PI -PPR analysis. The magnetic field plot is displayed:



Current Density

Double-click **Current Density** in the Results list of PI-PPR analysis.



See Also

- Setup Options for Customizing Results

SIPro Analysis

SIPro Analysis

Contents

- Tutorial-Performing Power Aware SI Analysis
- Creating a Power Aware SI Analysis Setup
- Viewing Power Aware SI Analysis Results

Tutorial-Performing Power Aware SI Analysis

In this tutorial, you will learn how to create a Power Aware SI analysis setup, run the analysis, and view results. The example workspace contains a Samsung DDR4 UDIMM memory card. The design files are downloaded from JEDEC (www.jedec.com) and used for demonstration purposes. The example design consists of a 6 layer board with a single power rail for core and I/O buffers, as shown in the following figure:

You can view the nets, components instances, and individual nets in the navigator window. The main power net passes from the conductor at the bottom of the design to the devices at the top. GND nets are present on the design. The substrate pack consists of routing on the top and bottom layer. The power and GND nets are on layer 3 and 4. Before performing the PIPro analysis, you need to know the names of the power and GND nets.

In this example design, the goal is to analyze the signal behavior of the critical line, including the effect of the GND metallization.

The goal of SI analysis is to provide an S-parameter model for the DQR15 line.

Example Workspace

```
examples/HSD/SIPro_PIPro_Getting_Started_Example_wrk.7zads
```

Objective

- Set up a Power Aware SI analysis
- Run the Power Aware SI analysis
- View Power Aware SI results (S-parameter and Generate Schematic)

Opening the Example Design

Open the *SIPro_PIPro_Getting_Started_Example_wrk.7zads* design. The SI analysis setup data for a specific design is stored in a cell view of the “SIPro/PIPro Setup”.

To create a **sipiSetup** view, open the layout and select **Tools** > **SIPro/PIPro** > **Open Setup**.

If the **sipiSetup** view already exists, you can open it with a double-click in the ADS Main window.

The SIPro/PIPro Setup window is displayed.

Creating a Power Aware SI Setup

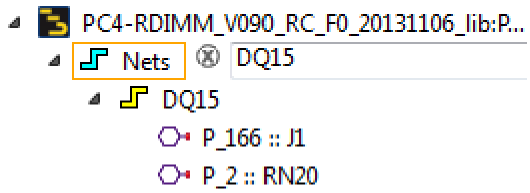
In this example, you will create a new SI analysis setup. To create a new setup, right-click **Analyses** and then select **New SI Analysis**.

Defining Ports

For an SI analysis, you need to define ports that will be the ports of the S-parameters calculated by the tool. In this example, you will define two ports, one on each side of the single DQR15 line.

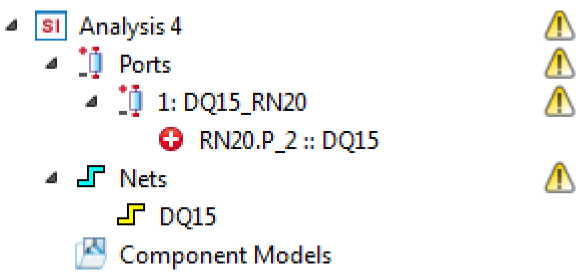
To define a port:

1. Select the **DQ15** line in the Nets tree.

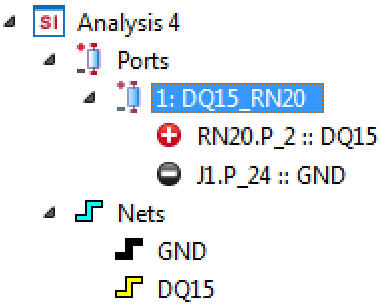
   

   In this example, you will use pins 6 and C2 to define the ports.

2. Drag and drop pin **RN20** on the ports part of the tree.

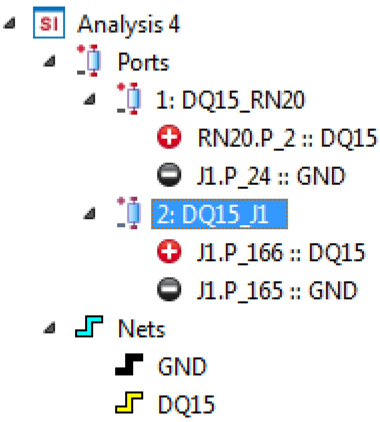
   

   By hovering over the exclamation mark, you will see that the GND reference for the port is not defined. You can either manually select a GND pin and drag it to the port. Alternatively, you can automatically find the closest GND pin by right-clicking **1: port** and then selecting **Add Nearby Pins To Minus Terminal**, which will add **J1.P24** as the minus reference to the port:



NOTE Two nets were automatically added to the list of nets: GND and DQ15. The metallization associated with these nets will be included in the Power Aware SI simulation.

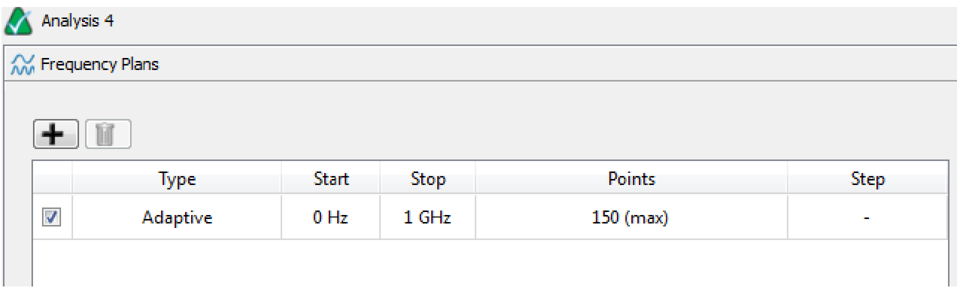
3. Drag and drop pin **J1** to the I/O Ports part of the tree.
4. Right-click **2: DQ15_J1** and select **Add Nearby Pins To Minus Terminal**, which will add **J1.P_165** as the minus reference to the port:



Defining Options

To define options:

1. Double-click **Options** in the SI Analysis list.
2. Double-click in the **Points** field of the **Adaptive** frequency plan and type **150**.



3. Click the **Options** tab.
4. Accept the default **Resolution** and **Arc Resolution** value.
5. Optionally, you can model the GND metallization as ideal PEC by selecting the ‘Use Ideal Ground Approximation’ option, which will result in a faster overall simulation.
6. Click **Done**.

Running the Analysis

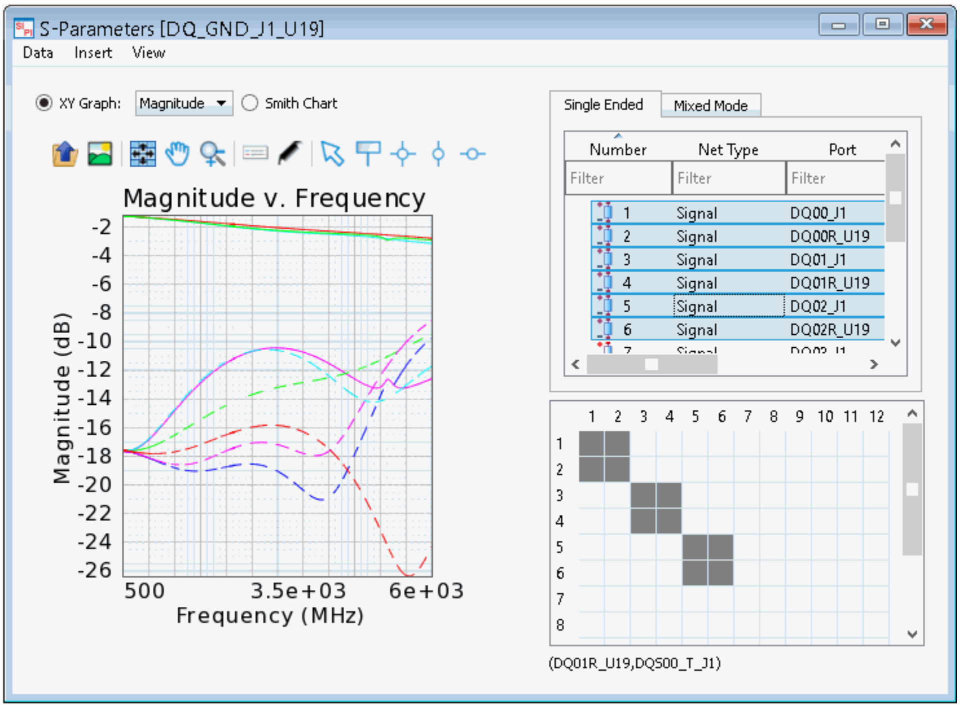
Save your SI analysis setup and double-click **Run** to start the SI analysis. The **Simulations** window is displayed and the SI analysis is initiated.

Viewing Results

After the analysis is complete, you can view results in the Results list. You can open the results during a simulation to viewing the intermediate results.

To view the S-parameters results:

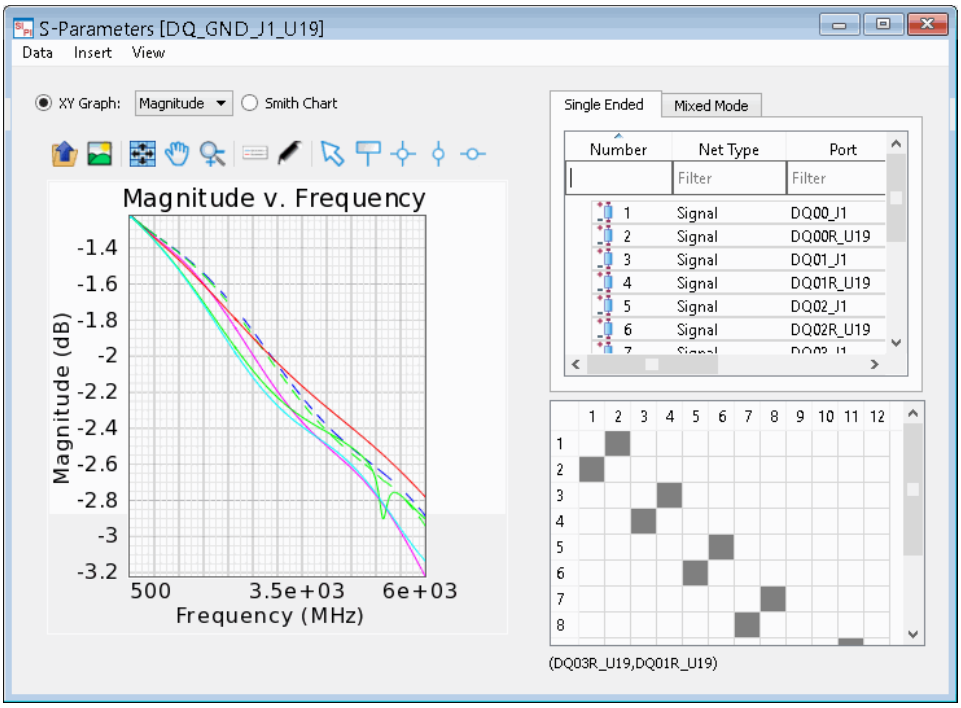
1. Double-click the **S-Parameters** option in the Results list.
2. Select the required nets in the **Single-Ended** section.
3. Right-click the selected nets and select **Add Return Loss**.



View TDR/TDT Results

To View TDR/TDT results:

1. Double-click the **TDR/TDT** option in the Results list.
2. Select the required nets in the **Single-Ended** tab.
3. Right-click the selected nets and select **Add TDR**.



Show Me How Do I Perform a Power Aware SI Analysis

Video: How to Perform a Power Aware SI Analysis

Creating a Power Aware SI Analysis Setup

A Power Aware SI analysis enables you to compute the S-parameters over a broad frequency range for one or more critical lines defined via I/O ports in the setup.

Creating a New Power Aware SI Analysis Setup

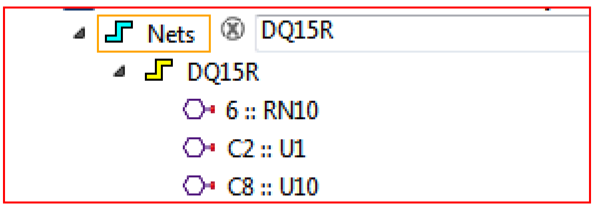
If you want to create a new setup, right-click **Analyses** and then select **New SI Analysis**.

Alternatively, you can copy an existing analysis setup to create a new one.

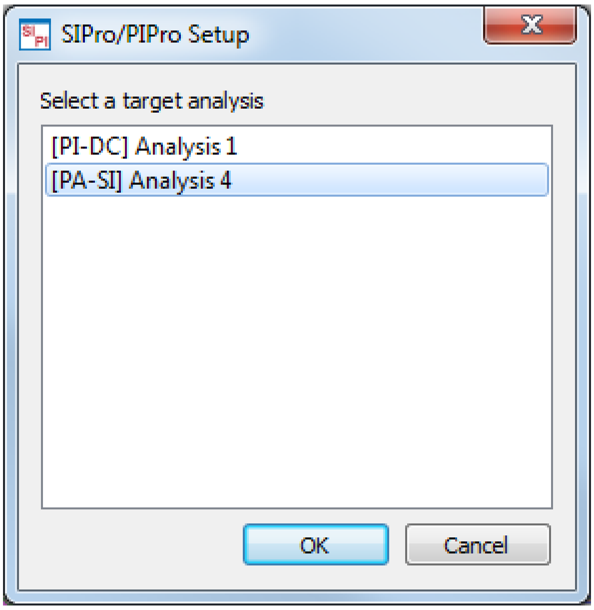
Selecting Nets

To add nets to the SI analysis node in the SIPro/PIPro setup:

1. Select the required nets in the Project panel:



2. Right-click the nets and select **Add to Analysis**.
3. Select **PA-SI Analysis** as the target analysis.



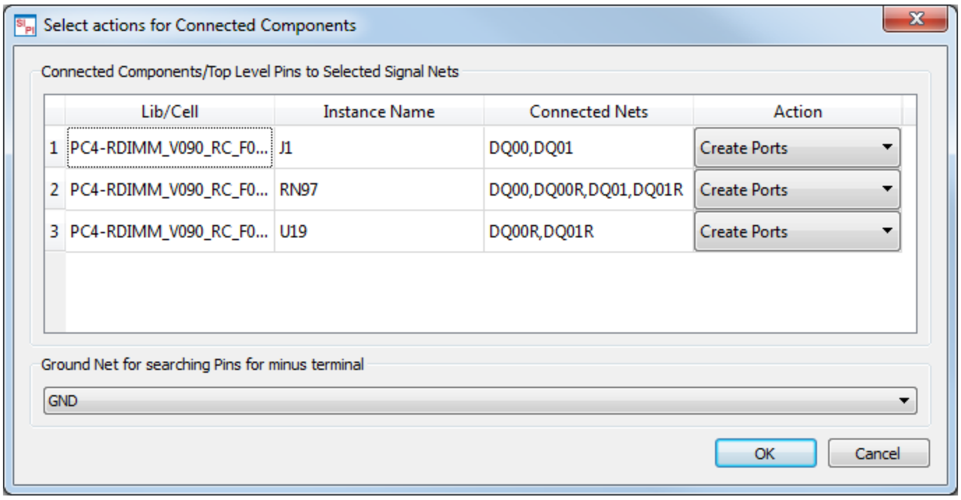
4. Click **OK**. The selected nets are added to the SI Analysis list.

Defining I/O Ports

The I/O ports define the + and – terminals for the S-parameter ports in the Power Aware SI simulation.

To define an I/O Port:

1. Select the required nets in the **Nets** list of the SI Analysis node.
2. Right-click the selected nets and select **Create Ports or Component Model Groups**.

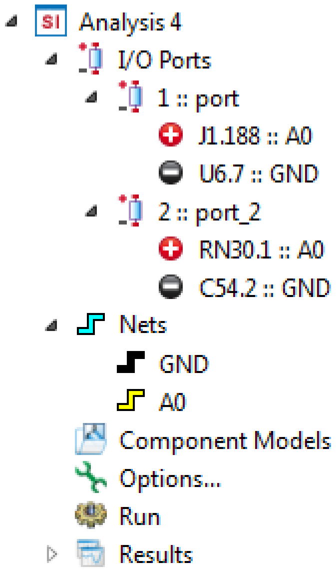


3. To create a port, accept the default selection, **Create Ports**.
4. Click **OK**.

Alternatively, you can drag and drop pins. To define ports:

1. Drag a pin connected to the net that you want to model to the I/O Ports in the project panel. This will create a new I/O port with only the + connection defined.

2. To complete the definition of this I/O Port, drag a nearby pin connected to a GND or PWR net to the I/O Port definition, or alternatively automatically connect the closest GND pin to the – terminal of the port by right-clicking the Port and selecting **Add Nearby Pins To Minus Terminal**.



NOTE This operation will automatically populate the Nets list with the nets connected to the + and – terminal.

Defining Component Models

Optionally, you can add components (series components such as resistors or decoupling capacitors) in the SI analysis.

To define a component model:

1. Select the component group, which you want to analyze, in the Components list.
2. Drag and drop the component group on **Component Models**. The instances will be grouped by component name.
3. Define the electrical model parameters by using one of the following ways:
   a. Double-click the component instance to specify the electrical model parameters for that instance.
   b. Alternatively, you can open the Vendor Parts database by selecting **Tools** > **Vendor Parts DB**. Drag and drop the appropriate vendor part onto the component instance node (single model assignment) or onto the component model group node (group model assignment). For more information, see Using Vendor Parts DB Browser.

Defining Options

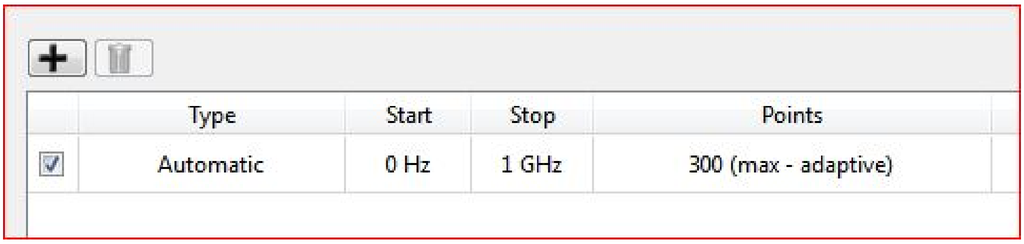
To define options, double-click Options in the SI Analysis list in the project panel. It provides the following options:

Option	Description
Resolution	Points closer than the specified resolution will be considered equal during the simulation

Option	Description
Arc Resolution	Value used to discretize circles and arcs during the simulation
Custom Target Mesh Size	Mesh size used to generate the global 3D mesh during simulation
Use Ideal Ground Approximation	When enabled, this option specifies that all Ground nets are considered as ideal shorts. The power rail is often the dominant contributor to the voltage drop. This option provides you the capability to analyze this dominant factor in less simulation time.
Use Optimized Via Modeling	Use via modeling settings optimized for simulation performance.
Model Signal Return Currents in Coplanar Ground	Ground metallization close to signal metallization is included in the signal modeling during simulation

Defining Frequency Plans

Click the **Frequency Plans** tab to specify a plan.You can use the Automatic, Linear, Logarithmic and Adaptive frequency plan. The Automatic uses a optimal combination of Adpative and Linear sweeps to cover the frequency range. The Start frequency can be 0 Hz, the simulator will assume 1 kHz when 0 Hz is specified.



When using an Adaptive sweep for electrically long lines on a complex board, a large number of frequency points may be required to reach convergence, specifically when a large number of signal lines is being analyzed. A linear frequency plan setup is advised in this case.

To capture the dynamic behavior of a signal line when using a Linear sweep, you will need 10 frequency points per wavelength, so in case the length of the signal line is 5 wavelengths, you will need 50 frequency points. This is typically sufficient when using the SiPro generated S-parameter model in e.g. a transient or channel simulation.

Saving the Setup

Save your SI analysis setup and double-click **Run** to start the SI analysis.

Viewing Power Aware SI Analysis Results

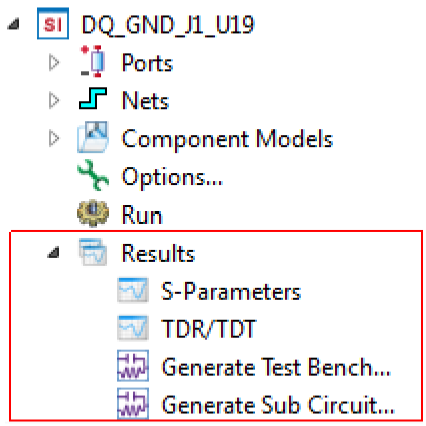
After creating an SI analysis setup, you can run the analysis in the SIPro/PIPro Setup window. For an SI analysis, the following results are displayed:

- S-Parameters
- TDR/TDT
- Generate Sub Circuit

Running a Power Aware SI Analysis

To run an SI analysis and view results:

1. Save your SI analysis setup.
2. Double-click **Run** to start the SI analysis. The Simulations window is displayed, which allows you to monitor and manage simulations.
3. Expand the Results list:

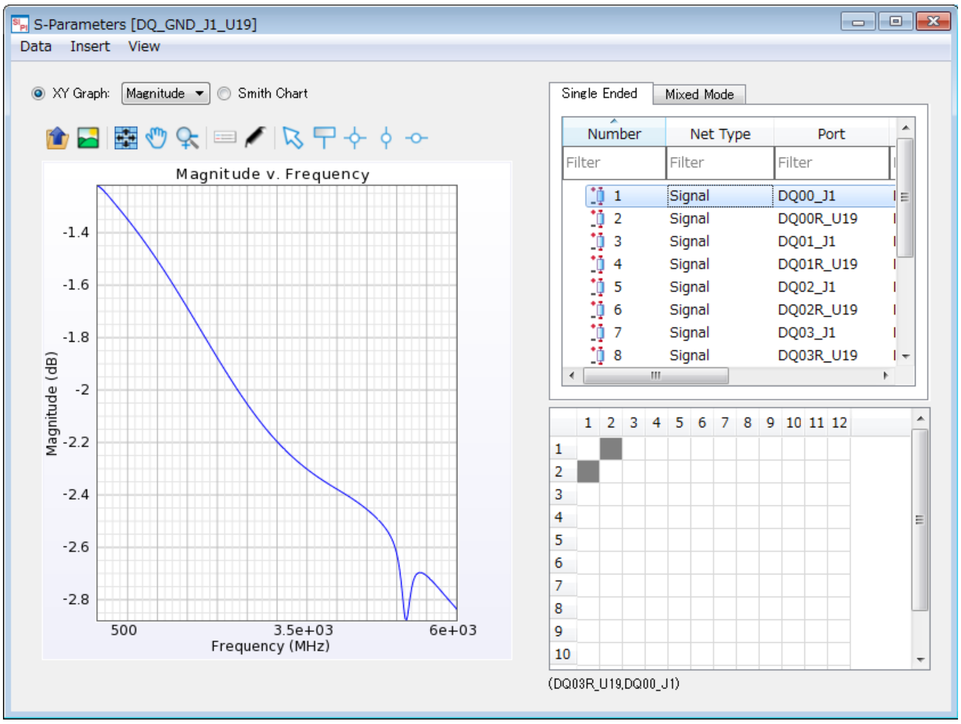


You can view S-parameters and generate a Schematic.

S-Parameters

To view the S-parameters results:

1. Double-click the **S-Parameters** option in the Results list. The S-Parameters window is displayed.
2. Select the required nets in the **Single-Ended** section.
3. Right-click the selected nets and select the required option such as, **Add transmission**.



Using the S-Parameters Results Window

The S-Parameters window consists of the following menus:

- Data
- Insert
- View

Data

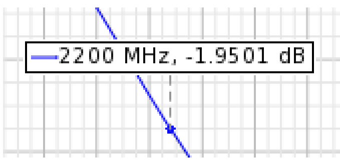
The Data menu provides the following options:

- **Export** : Exports the currently displayed plot to a text file.
- **Save Plots as New Graph**: Copies the current graph including plots and markers. It is saved in the **Graphs** node in the project tree.

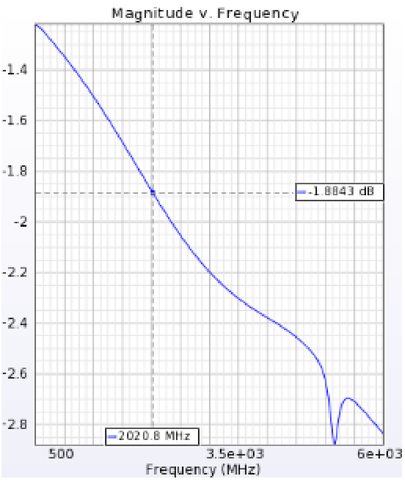
Insert

The Insert menu provides the following options:

- **Point Marker**: Insert a point marker in the graph.



- **Crosshair Marker**: Insert a crosshair marker.



- **Horizontal Marker**: Insert a horizontal marker.
- **Vertical Marker**: Insert a vertical marker.

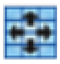
View

The View menu provides the following options:

- **Pan**: Left-click on the graph and move mouse pointer to pan.
- **Zoom**: Left click on the graph and move mouse pointer to create a rectangle to zoom.
- **Zoom to Extents**: Zoom in or out to current plots so that they can fit into the graph.
- **Export Image**: Export plot image to a file.

Toolbar Options

The S-Parameters window provides several toolbar options for customizing the graph.

Icon	Description
	The Export Data icon enables you to export the currently displayed plot to a text file.
	The Export Image icon enables you to export the plot image to a file.
	The Zoom to Extents icon enables you to zoom in or out to current plots so that they can fit into the graph.
	The Pan icon enables you to move mouse pointer to pan.
	The Zoom Tool icon enables you to move mouse pointer to create a rectangle to zoom.
	

Icon	Description
	The Legend Visible icon enables you to show or hide the legends of the plots.
	The Selection tool icon enables you to select items in a plot.
	The Point Marker Tool insert a point marker in the graph.
	The Crosshair Marker icon inserts a crosshair marker.
	The Horizontal Marker icon inserts a horizontal marker.
	The Graph Properties icon displays the graph properties pane. You can change graph appearance, and axis controls.

Graph Type Selector

You can set the following **Graph Type Selector** options to select a graph type:

- **XY Graph Magnitude**: Displays magnitude of the plots in the XY rectangular graph.
- **XY Graph Phase**: Displays phase of the plots in the XY rectangular graph.
- **Smith Chart**: Displays plots in the Smith chart.

Single Ended Tab

In the Single Ended tab, you can view the port properties, change the port impedance, and manipulate plots by right-clicking on each row in the table.

Port Property Table

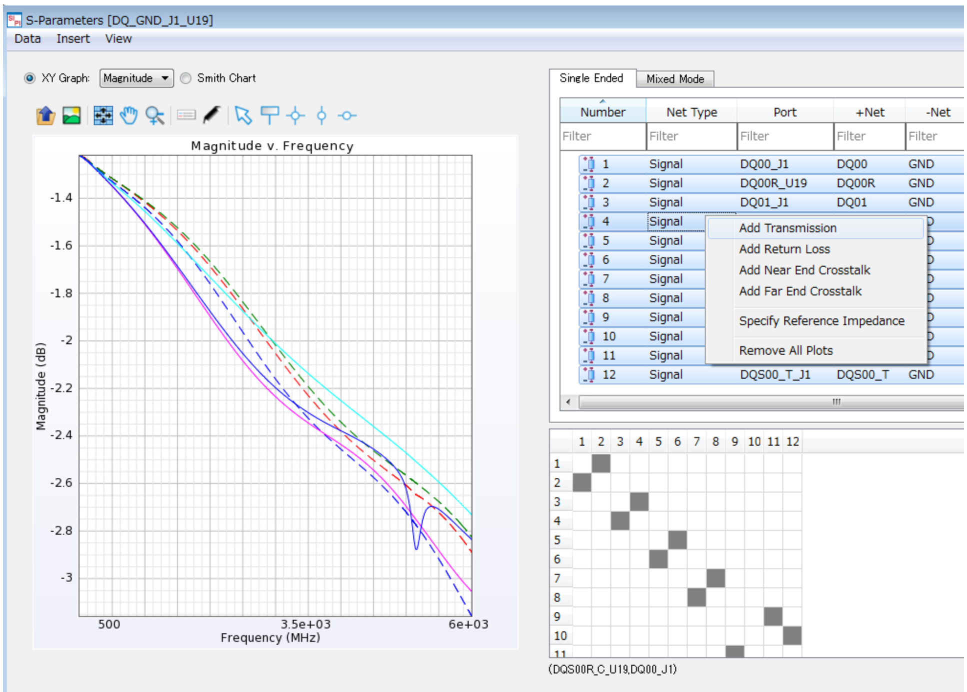
The Port Property Table shows various port properties in a table.

Number	Net Type	Port	+Net	-Net	Instance	Z
Filter	Filter	Filter	Filter	Filter	Filter	Filter
1	Signal	DQ00_J1	DQ00	GND	J1	50 ohm
2	Signal	DQ00R_U19	DQ00R	GND	U19	50 ohm
3	Signal	DQ01_J1	DQ01	GND	J1	50 ohm
4	Signal	DQ01R_U19	DQ01R	GND	U19	50 ohm
5	Signal	DQ02_J1	DQ02	GND	J1	50 ohm
6	Signal	DQ02R_U19	DQ02R	GND	U19	50 ohm
7	Signal	DQ03_J1	DQ03	GND	J1	50 ohm
8	Signal	DQ03R_U19	DQ03R	GND	U19	50 ohm
9	Signal	DQS00R_C_U19	DQS00R_C	GND	U19	50 ohm
10	Signal	DQS00R_T_U19	DQS00R_T	GND	U19	50 ohm
11	Signal	DQS00_C_J1	DQS00_C	GND	J1	50 ohm
12	Signal	DQS00_T_J1	DQS00_T	GND	J1	50 ohm

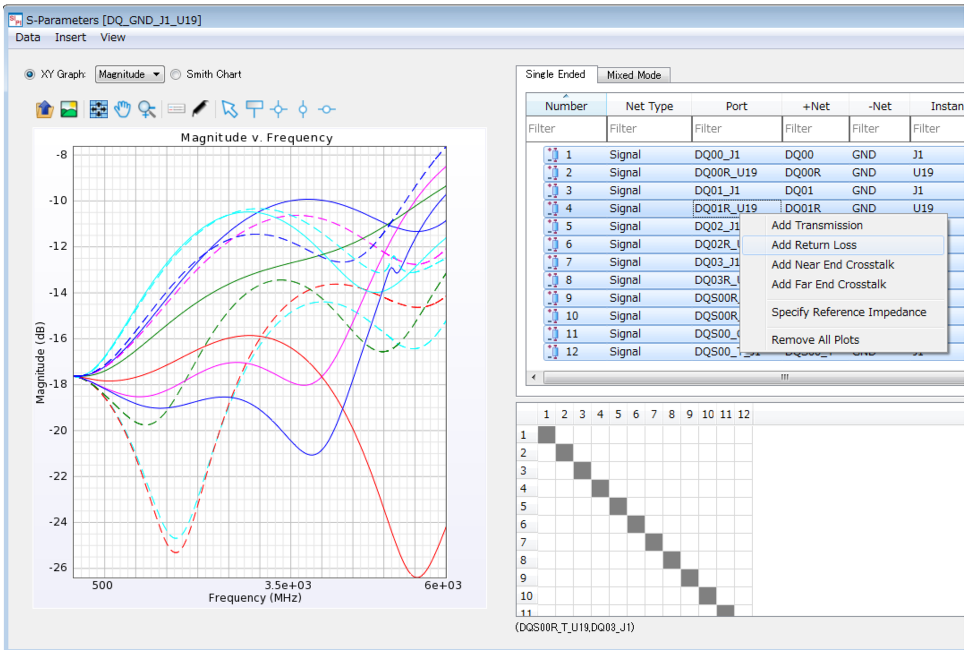
- **Number:** Specifies to the S-parameter index.
- **Net Type**: Specifies whether the port's plus terminal connects to Signal nets or Power nets
- **Port**: Port name that you define in the analysis tree
- **+Net**: Specifies the net name(s) that is connected to the plus terminal of the port.
- **-Net**: Specifies the net name(s) that is connected to the minus terminal of the port.
- **Instance**: Specifies the instance name(s) with which a port connected. For top level pins, this property becomes blank.
- **Z**: Specifies the port impedance. This property is editable. To edit, double-click the cell or select Specify Reference Impedance context menu.

You can perform the following tasks by right-clicking a port and selecting the required option:

- **Viewing the Transmission plot** : Right-click the selected port and then select **Add transmission** to display the transmission characteristic(s) of selected port(s) to the graph. This feature will automatically find the other end of connected link of the selected ports. This feature is convenient to check transmission characteristics for the multiple lines.



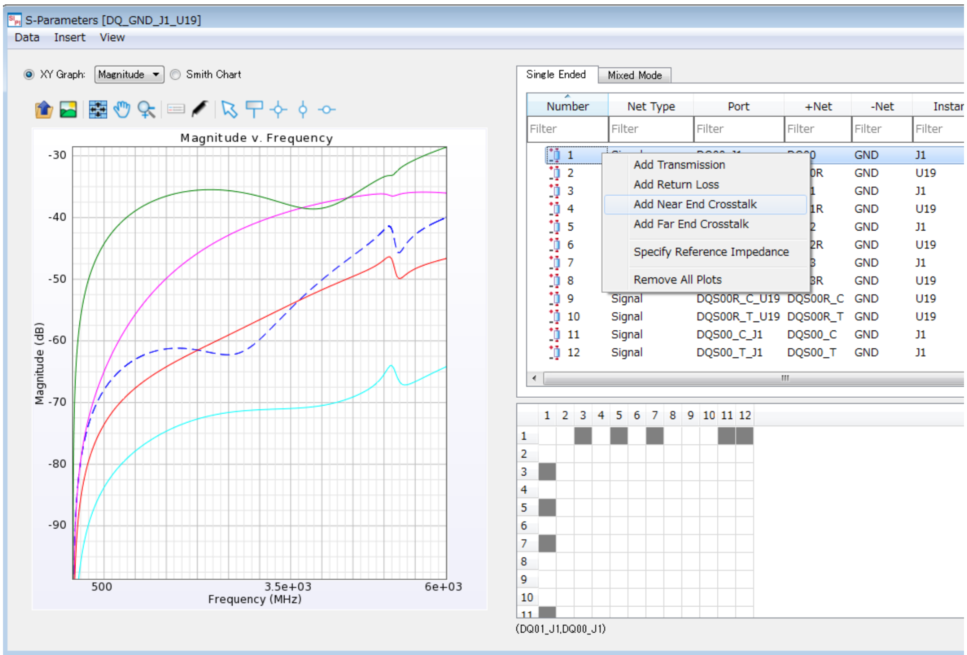
- **Viewing the Return Loss**: Right-click the selected port(s) and select **Add return loss** to display the return plot(s) of selected port(s) to the graph.



- **View the Insertion Loss**: Right-click a net and then select **Add Insertion Loss** to display the insertion loss between the selected two ports to the graph.

> **NOTE**
>
> **Insertion loss** here does not necessarily mean "transmission". It means loss between any of two ports.

- **View Near End Crosstalk**: Right-click a port and then select **Add Near End Crosstalk** to view the near end crosstalk (s)of selected port(s) in the graph. This feature will automatically find all near end cross talk of the selected ports with using **Instance** information.



- **Add Far End Crosstalk** : Add far end crosstalk(s) of selected port(s) to the graph. This feature will automatically find all far end cross talk of the selected ports with using **Instance** information.
- **Specify Reference Impedance** : Set preferred reference impedance to the selected ports.
- **Remove All Plots** : Remove all plots from the graph.

Mixed Mode Tab

In the Mixed Mode tab, you can define differential pair of ports and add mixed mode plots in the graph in addition to most of what you can do in the Single Ended tab.

Number	Signal	Net Type	Port	+Net	-Net	Instance	Z
Filter	Filter	Filter	Filter	Filter	Filter	Filter	Filter
1	SE	Signal	DQ00_J1	DQ00	GND	J1	50 ohm
2	SE	Signal	DQ00R_U19	DQ00R	GND	U19	50 ohm
3	SE	Signal	DQ01_J1	DQ01	GND	J1	50 ohm
4	SE	Signal	DQ01R_U19	DQ01R	GND	U19	50 ohm
5	SE	Signal	DQ02_J1	DQ02	GND	J1	50 ohm
6	SE	Signal	DQ02R_U19	DQ02R	GND	U19	50 ohm
7	SE	Signal	DQ03_J1	DQ03	GND	J1	50 ohm
8	SE	Signal	DQ03R_U19	DQ03R	GND	U19	50 ohm
9	D	Signal	DQS00R_C_U19|DQS00R_T_U19_D	DQS00R_C	DQS00R_T	U19	100
10	C	Signal	DQS00R_C_U19|DQS00R_T_U19_C	DQS00R_C DQS00R_T	GND	U19	25
11	D	Signal	DQS00_C_J1|DQS00_T_J1_D	DQS00_C	DQS00_T	J1	100
12	C	Signal	DQS00_C_J1|DQS00_T_J1_C	DQS00_C DQS00_T	GND	J1	25

Port Property Table

In the Mixed Mode tab, **Signal** column is added to the ones for Single Ended table.

- **Signal**: Shows the type of the port.
  - **SE** stands for Single Ended
  - **D** stands for Differential
  - **C** stands for Common.

Context menus

In mixed mode tab, some context menus are added and some context menus are removed comparing to the one for Single Ended tab.

Context menus only for Mixed Mode tab

- **Make Differential Pairs**: Make differential pairs for selected ports. This context menu works in two different ways depending on how many ports you select in the table.
  - If you select two ports in the table it means to create an differential pair for the selected two ports explicitly. If selected ports are already a part of other differential pair that differential pair will be broken before creating new differential pair.
  - If you select more than two ports in the table it means to create multiple differential pairs for selected ports. In this case differential pairs will be recognized and created automatically based on net names and instance name with using predefined Differential Pair Recognition Rule. (See more detail here).
  - In below example picture, selecting 4 ports and using Make Differential Pair for them will create two pairs as follows.

Number	Signal	Net Type	Port	+Net	-Net	Instance
Filter	Filter	Filter	Filter	Filter	Filter	Filter
1	SE	Signal	DQ1_Rx	DQ1	Gnd	Rx
2	SE	Signal	DQ1_Tx	DQ1	Gnd	Tx
3	SE	Signal	DQ1#_Rx	DQ1#	Gnd	Rx
4	SE	Signal	DQ1#_Tx	DQ1#	Gnd	Tx

  - Port1 and 3 as a differential pair because these plus nets are DQ1 and DQ1# which are recognized as differential pair by the recognition rule and also these are on the same instance Rx.
  - Port2 and 4 as a differential pair because these plus nets are DQ1 and DQ1# which are recognized as differential pair by the recognition rule and also these are on the same instance Tx.

Number	Signal	Net Type	Port	+Net	-Net	Instance
Filter	Filter	Filter	Filter	Filter	Filter	Filter
1	D	Signal	DQ1_Rx|DQ1#_Rx_D	DQ1	DQ1#	Rx
2	D	Signal	DQ1_Tx|DQ1#_Tx_D	DQ1	DQ1#	Tx
3	C	Signal	DQ1_Rx|DQ1#_Rx_C	DQ1 DQ1#	Gnd	Rx
4	C	Signal	DQ1_Tx|DQ1#_Tx_C	DQ1 DQ1#	Gnd	Tx

- **Break Differential Pairs** : Break existing pairs for selected ports.

Context menus not available for Mixed Mode tab

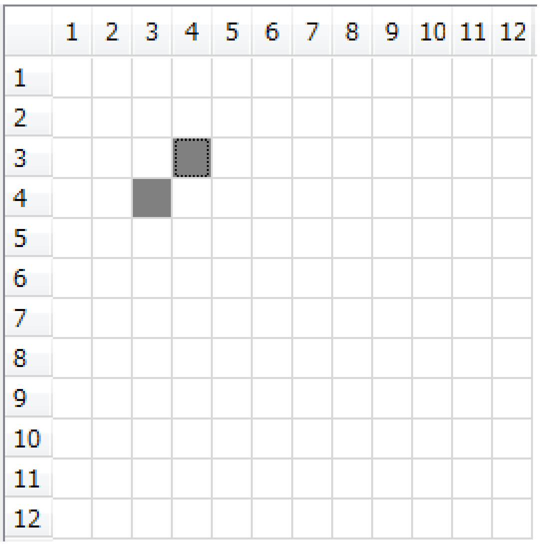
- **Add Near End Crosstalk**
- **Add Far End Crosstalk**

View Mixed Mode S-parameters

- **View Return Loss :** Use **Add Return Loss** context menu on desired ports.
- **View Insertion Loss** : Use **Add Insertion Loss** context menu on desired two ports that you want to view insertion loss in between. (Note: term of "Insertion Loss" doesn't necessarily mean insertion loss of physically connected link.)

Matrix Selector

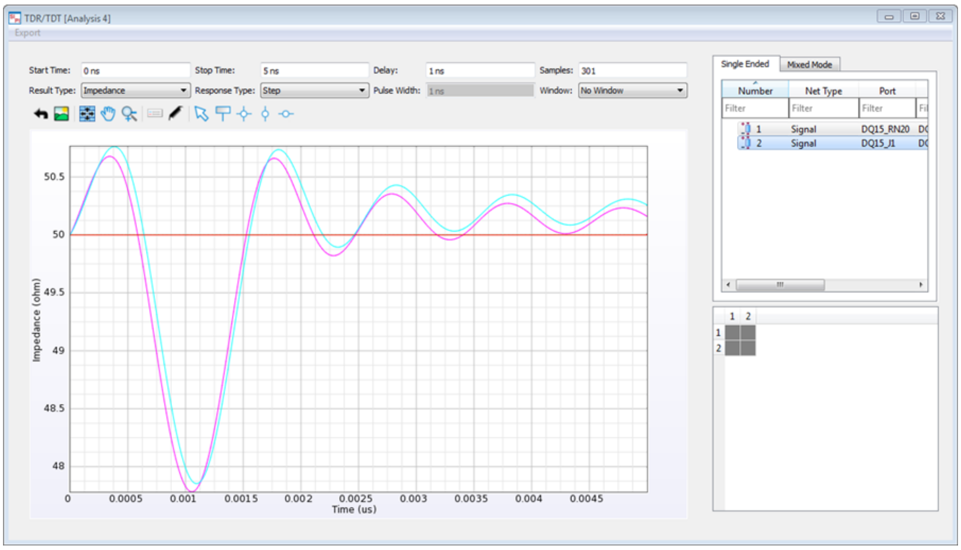
Matrix Selector allows you to select add/remove plots based on S-parameter index.



View TDR/TDT Results

To view TDR/TDT results:

1. Double-click the **TDR/TDT** option in the Results list.
2. Select the required nets in the **Single-Ended** section.
3. Right-click the selected nets and select the required option such as, **Add TDR**.



Using the TDR/TDT Results Window

The TDR/TDT window consists of the following menus:

- Export

Export

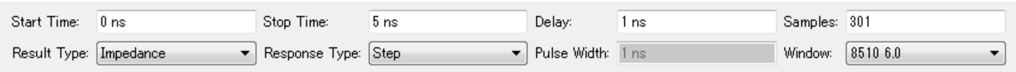
The Export menu provides the following options:

- **Save Plots as New Graph**: Copies the current graph including plots and markers. It is saved in the **Graphs** node in the project tree.

Toolbar Options

The TDR/TDT window provides several toolbar options for customizing the graph. See **Toolbar Options** section in the Using the **S-Parameters Results Window** for more detail.

TDR/TDT Setup Options



Start Time : Specify start time.

Stop Time : Specify stop time.

Delay : Specify delay value.

Samples : Specify number of data samples in time domain.

Result Type : Specify Y-axis unit in either Impedance or Voltage.

Response Type : Specify response type either Step or Pulse.

Pulse Width : Specify pulse width for Pulse Response Type. This value is ignored in Step Response Type.

Window : Specify window type

Single Ended Tab

In the Single Ended tab, you can view the port properties, change the port impedance, and manipulate plots by right-clicking on each row in the table.

Number	Net Type	Port	+Net	-Net	Instance	Z
Filter	Filter	Filter	Filter	Filter	Filter	Filter
1	Signal	DQ00_J1	DQ00	GND	J1	50 ohm
2	Signal	DQ00R_U19	DQ00R	GND	U19	50 ohm
3	Signal	DQ01_J1	DQ01	GND	J1	50 ohm
4	Signal	DQ01R_U19	DQ01R	GND	U19	50 ohm
5	Signal	DQ02_J1	DQ02	GND	J1	50 ohm
6	Signal	DQ02R_U19	DQ02R	GND	U19	50 ohm
7	Signal	DQ03_J1	DQ03	GND	J1	50 ohm
8	Signal	DQ03R_U19	DQ03R	GND	U19	50 ohm
9	Signal	DQS00R_C_U19	DQS00R_C	GND	U19	50 ohm
10	Signal	DQS00R_T_U19	DQS00R_T	GND	U19	50 ohm
11	Signal	DQS00_C_J1	DQS00_C	GND	J1	50 ohm
12	Signal	DQS00_T_J1	DQS00_T	GND	J1	50 ohm

Port Property Table

The Port Property Table shows various port properties in a table. See **Port Property Table** section in the Using the **S-Parameters Results Window** for more detail of table contents.

You can perform the following tasks by right-clicking a port and selecting the required option:

- **Viewing the TDT plot** : Right-click the selected port and then select **Add TDT** to display the TDT characteristic(s) of selected port(s) to the graph. This feature will automatically find the other end of connected link of the selected ports. This feature is convenient to check transmission characteristics for the multiple lines.
- **Viewing the TDR plot** : Right-click the selected port and then select **Add TDR** to display the TDR characteristic(s) of selected port(s) to the graph.
- **Specify Reference Impedance** : Set preferred reference impedance to the selected ports.
- **Remove All Plots** : Remove all plots from the graph.

Mixed Mode Tab

In the Mixed Mode tab, you can define differential pair of ports and add mixed mode plots in the graph in addition to most of what you can do in the Single Ended tab.

Number	Signal	Net Type	Port	+Net	-Net	Instance	Z
Filter	Filter	Filter	Filter	Filter	Filter	Filter	Filter
1	SE	Signal	DQ00_J1	DQ00	GND	J1	50 ohm
2	SE	Signal	DQ00R_U19	DQ00R	GND	U19	50 ohm
3	SE	Signal	DQ01_J1	DQ01	GND	J1	50 ohm
4	SE	Signal	DQ01R_U19	DQ01R	GND	U19	50 ohm
5	SE	Signal	DQ02_J1	DQ02	GND	J1	50 ohm
6	SE	Signal	DQ02R_U19	DQ02R	GND	U19	50 ohm
7	SE	Signal	DQ03_J1	DQ03	GND	J1	50 ohm
8	SE	Signal	DQ03R_U19	DQ03R	GND	U19	50 ohm
9	D	Signal	DQS00R_C_U19|DQS00R_T_U19_D	DQS00R_C	DQS00R_T	U19	100
10	C	Signal	DQS00R_C_U19|DQS00R_T_U19_C	DQS00R_C DQS00R_T	GND	U19	25
11	D	Signal	DQS00_C_J1|DQS00_T_J1_D	DQS00_C	DQS00_T	J1	100
12	C	Signal	DQS00_C_J1|DQS00_T_J1_C	DQS00_C DQS00_T	GND	J1	25

Port Property Table

In the Mixed Mode tab, **Signal** column is added to the ones for Single Ended table.

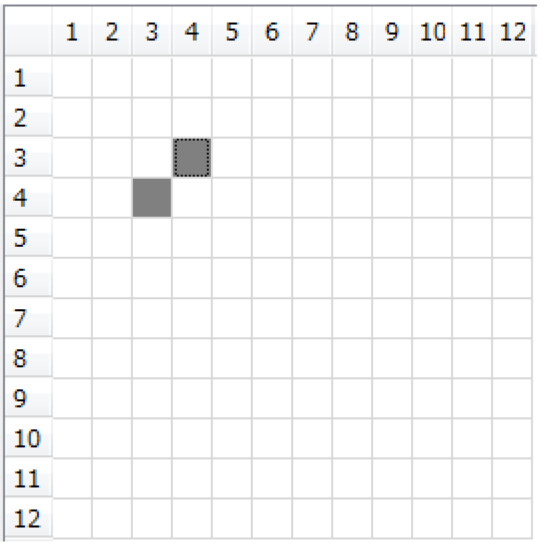
- **Signal**: Shows the type of the port.
  - **SE** stands for Single Ended
  - **D** stands for Differential
  - **C** stands for Common.

In mixed mode tab, following context menus are added comparing to the one for Single Ended tab.

- **Make Differential Pairs**: Make differential pairs for selected ports. See the same section in the **Using the S-Parameters Results Window.**
- **Break Differential Pairs** : Break existing pairs for selected ports. See the same section in the **Using the S-Parameters Results Window.**

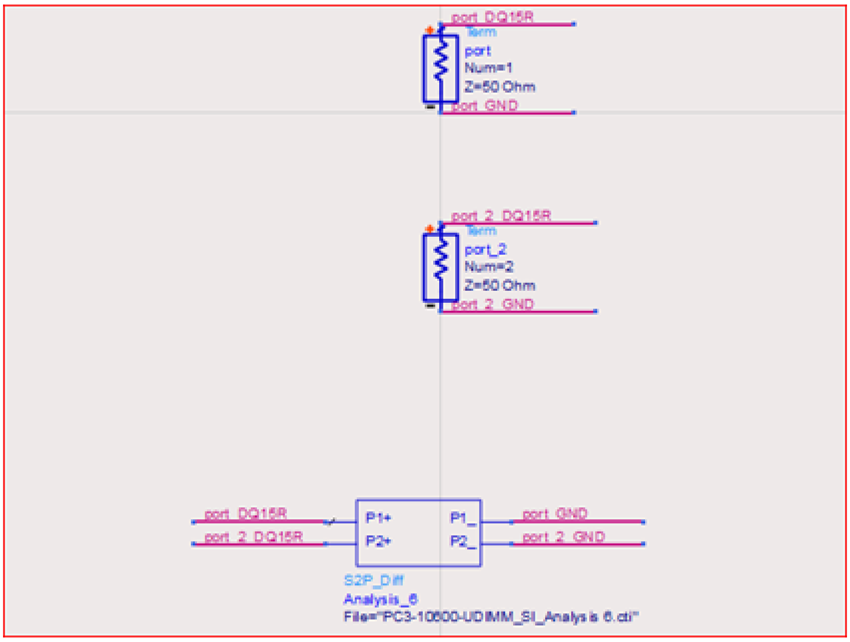
Matrix Selector

Matrix Selector allows you to select add/remove plots based on S-parameter index.



Generate Sub Circuit

Double-click **Generate Sub Circuit** in the Results list. A Schematic window is displayed, as shown in the following figure:



Installing the Deprecated SI and PI Analysis Addon

WARNING The legacy SI/PI analyzer is still available, but is a deprecated solution. Please use SIPro or PIPro instead, see Getting Started with SIPro and PIPro. This addon will no longer be available in the future major releases of ADS.

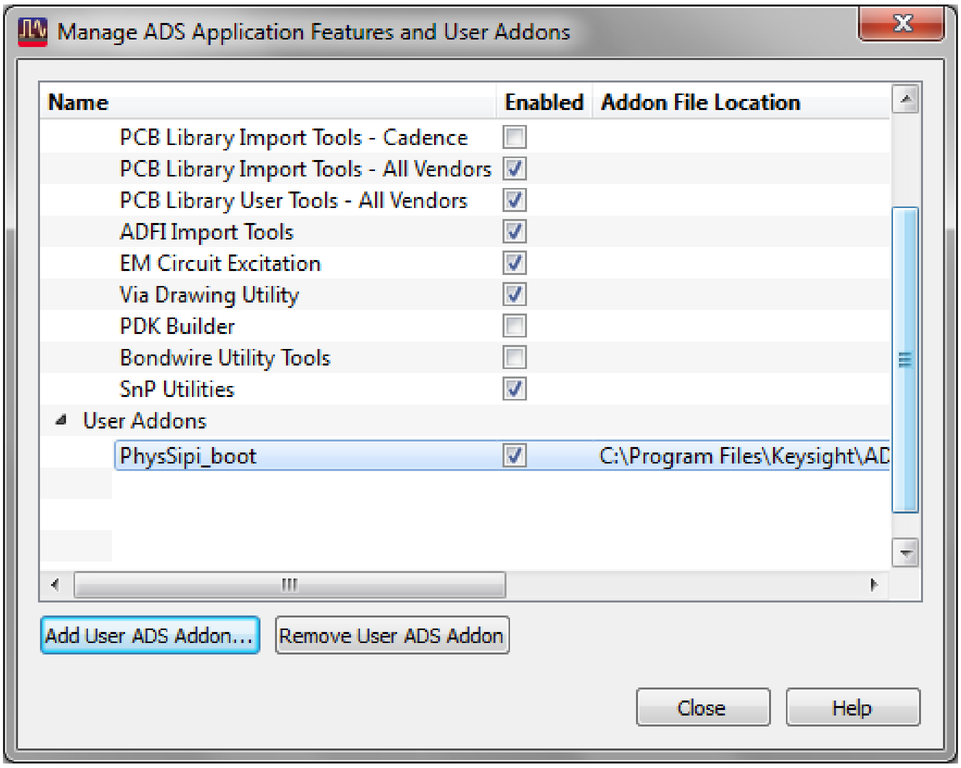
Installing the Deprecated SI and PI Analysis Addon

The legacy SI/PI analyzer is available as an addon. The SI/PI Analyzer addon assists in setting up SI or PI EM and circuit simulations. This addon consists of two parts:

- Setup wizard: The setup wizard guides you through the process of creating a new cell that contains only selected nets (physical interconnects) to be analyzed. The output of the setup wizard is a cell that is ready for EM analysis.
- Analysis guide: The analysis guide assists in setting up typical SI or PI circuit simulation testbench. The output of the analysis guide is a circuit schematic, ready to be simulated, a data display template that captures relevant output figures. In addition, the analysis guide supports Momentum current visualization based on the results.

To install the SI and PI Analyzer addon:

1. Select **Tools** > **App Manager** in the ADS Main window.
2. Click **Add User ADS Addon**. The Add User Addon dialog box is displayed.
3. Click **Browse**.
4. Access the **$HPEESOF_DIR/ael_addons/sipi_deprecated/ael** folder.
5. Select the **PhysSipi_boot.atf** file and click **Open**.
6. Click **OK**. The PhysSipi_boot option is displayed in the User Addons list.
7. Select the **Enabled** check box for **PhysSipi_boot**, as shown in the following figure:

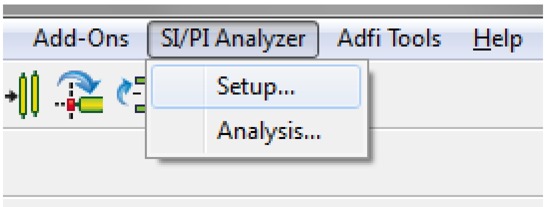


8. Click **Yes** in the message box.

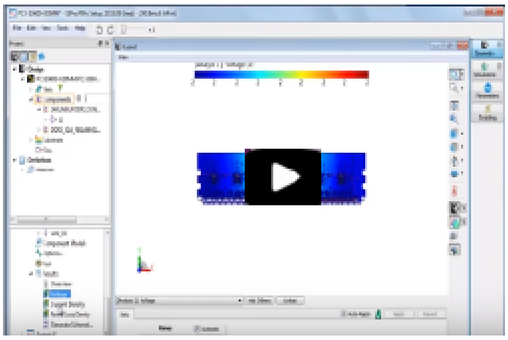
9. Click **OK** in the message box.

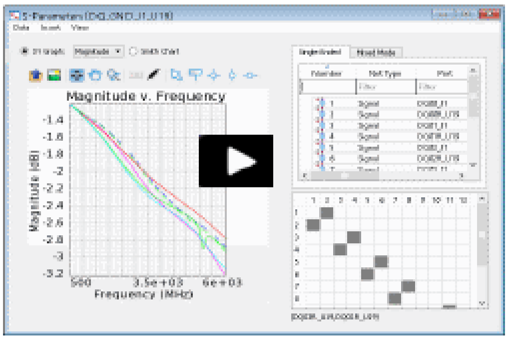
10. Click **Close** in the App Manager window.

11. Open a Layout window. The SI/PI Analyzer addon is displayed in the menu bar, as shown in the following figure:



SIPro/PIPro Videos



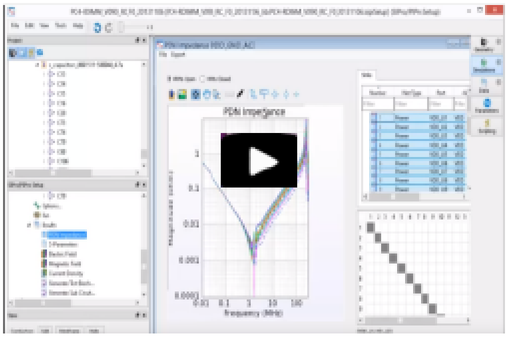


Perform a PI-DC Analysis

This video talks about how to perform a PI-DC analysis.

Perform a Power Aware SI Analysis

This video talks about how to perform a Power Aware SI analysis.



Perform a PI-AC Analysis

This video talks about how to perform a PI-AC analysis.