XtractIM Tutorial

Product Version 16.6
July 2014

Document Updated on: July 11, 2014

Table of Contents

Chapter 1

Introduction

Welcome to the XtractIM Tutorial. This manual is designed to give you a brief introduction to the XtractIM tool by providing real life examples and demonstrations so you can understand some of the basic concepts. The XtractIM tool is part of a complete suite of tools for package design.

System Requirements

Please refer to ***Installation Guide*** to check the system requirements.

How to Use This Guide

The XtractIM Tutorial provides demonstration examples and step-by-step instructions on how to get the desired results.

Go through the sections in each chapter in order. Perform all steps and study the examples.

Additional Documentation

In addition to this document, refer to the following documentation for additional information.

- ***XtractIM User's Guide*** describes in detail the features and functionality of XtractIM.
- ***Translators User's Guide*** describes translations from various types of board and package file formats to Sigrity's SPD format.

Conventions Used in This Guide

Convention	Use
Bold	GUI text, special names, terms (window names, buttons, menus, etc.)
Arial	Examples

CONVENTION	USE
>	Menu hierarchy

HOW TO CONTACT TECHNICAL SUPPORT

We are committed to helping you in using XtractIM. If you have any questions, contact the *Cadence Online Support*.

Chapter

2

Overview

This chapter introduces the functionality of XtractIM.

ABOUT XTRACTIM

XtractIM extracts the most common electrical models of IC packages according to IBIS (I/O Buffer Information Specification) as well as SPICE netlist of electrical models. XtractIM also supports electrical performance assessment on signal and power/ground distribution systems for packages. This feature helps you analyze the quality of the design.

These models can be used for system-level analysis including:

- Assessing the electrical performance of IC packages
- Drivers
- Interconnects
- Receivers

XtractIM can handle lead-frame, flip-chip packages and wirebond packages with 3D bonding wire profiles. The package can be single- or multi-die, and single- or stacked-BGA packages.

It can extract models of full packages or selected nets of a package.

Its interface is compatible with data files in various formats, including UPD, MCM, .BRD, .SIP, NA2, DSN, and SPD formats.

XtractIM has two modules in extraction: IBIS/RLGC and Optimized Broadband. XtractIM supports both net-based and pin-based RLC extraction for single or multi-die and single or stacked BGA packages.

Model Extraction

Model Extraction Mode includes

- RLC / IBIS Module
- Optimized Broadband Module

RLGC / IBIS Module

The RLGC / IBIS Module for single-die, single-BGA package provides with the capability to:

- Generate IBIS package pin RLC model
- Generate IBIS package RLC (Resistance, Inductance and Capacitance) matrix model with coupling between signal, power and ground nets
- Generate net length, DC_R, delay of each signal net
- Generate SPICE equivalent circuits of package RLGC models of different topologies (Pi or T), including coupling among signal, power and ground nets
- 2D and 3D display of RLC curves and distributions, including coupling between nets

The RLGC Module for Multi-Die, Stacked-BGA package provides the capability to:

- Generate DC_R of signal, power, and ground nets along each of circuit-to-circuit paths
- Generate self R, self-L, C of signal and power nets along each of circuit-to-circuit paths
- Generate mutual L and C of signal and power nets along each of the circuit-to-circuit paths
- Generate branch RL and total C of signal and power nets
- Generate SPICE equivalent circuits of package RLGC model including coupling among signal, power and ground nets

Optimized Broadband Module

The Optimized Broadband Module works for both single-die, single-BGA and multi-die stacked-BGA packages. It provides users with the following capabilities:

- Display of the original S-parameter and compact circuit model S-parameter
- Export the S-parameter model in Sigrity compact formats (BNP)
- Extract and display of S network parameters
- Optimized Broadband RLC circuit model extraction with options of selecting circuit models

Electrical Performance Assessment

Electrical Performance Assessment includes two systems.

- Power / Ground Distribution System
- Signal Distribution System

Power / Ground Distribution System

- Net loop inductance of Power Nets referring to each of the Ground Nets
- Broadband Impedance of each Power Net referring to its best Ground Net

- Plane IR drop, Plane current density and Via-current checking
- Per Pin resistance and inductance of power / ground net viewing from die-side

Signal Distribution System

- Trace layput checking: Impedance and strongest coupling of single-ended Net vs. section-by-section Net-length
- Net couplings: mutual inductance and capacitance, total near-ended crosstalk for each Net as a victim
- Broadband Insertion and Return Loss

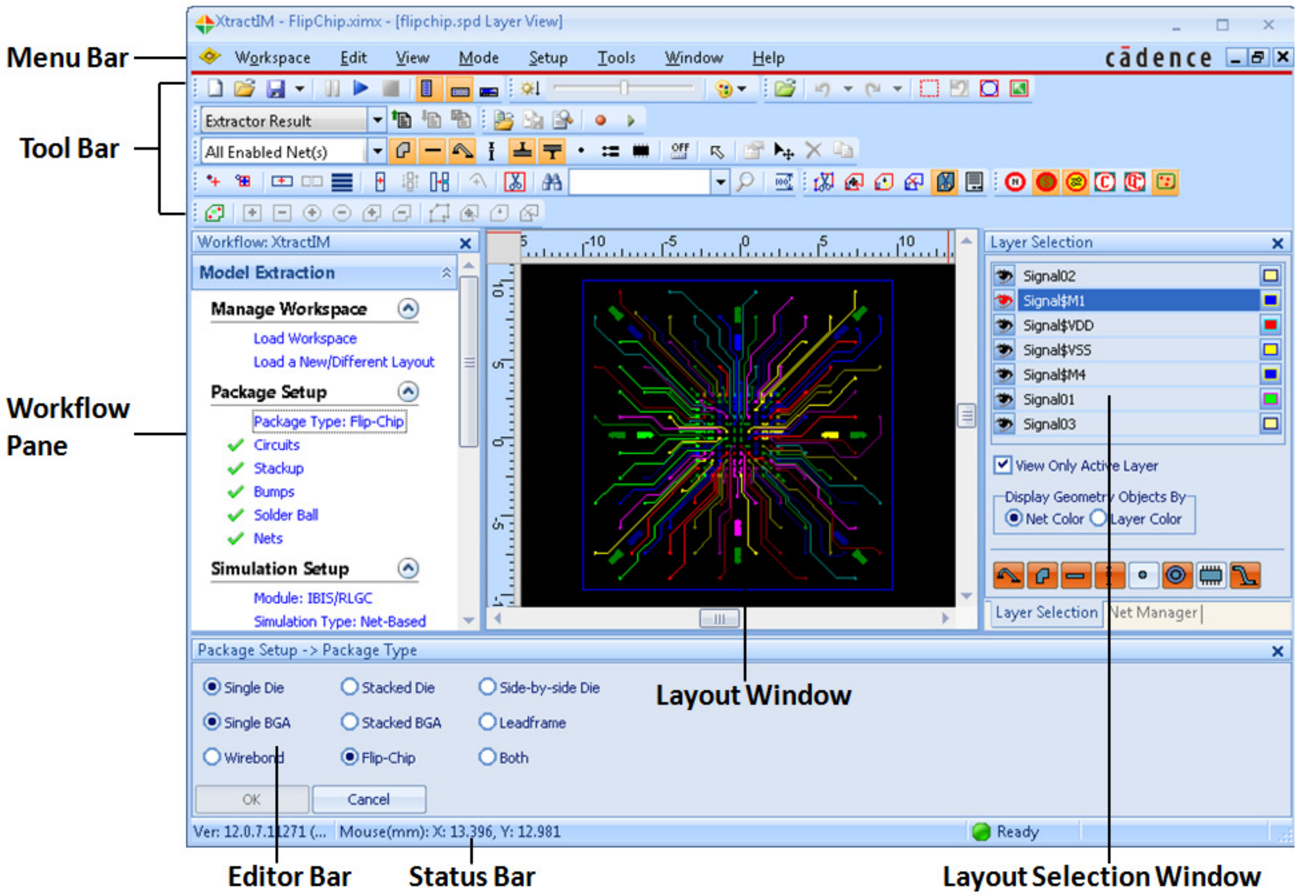
The XtractIM Window

The new workspace is made up of two main areas.

- **Layout Area** — Large area with a black background
- **Workflow Pane** — Left side of the screen

Within these two main areas you'll find toolbars, menus, and smaller, specialized panes. You'll use all the tools on the workspace to create a package simulation.

- **Editor Pane** — A spreadsheet. Users can easily input information into the pane for simulation setup or check simulation results in the pane
- **Layout Selection Window** — Controls the active layer, displayed layers and the object display on or off
- **Layout Window** — Where you edit your layout
- **Menu Bar** — The pull-down menus provide the commands you need to modify a design, set up a simulation, run a simulation, check results and report results
- **Status Bar** — Shows the version information, the current X-Y coordinates of the cursor and the simulation progress
- **Tool bar** — Gives you quick access to common XtractIM commands
- **Workflow Pane** — Lists all the workflow tasks. Tasks of the same type are sorted and listed together. When a task is clicked, details associated with that task appear in an Editor pane

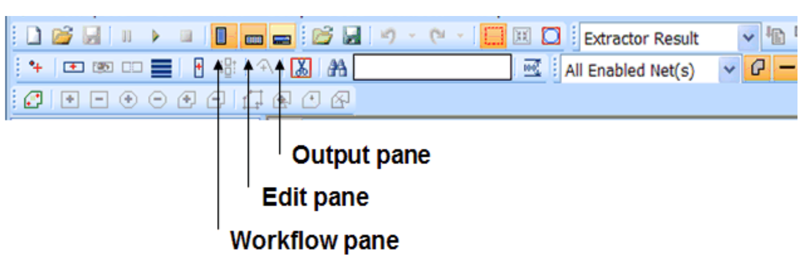


Using the Toolbar Icons

1. Click on **Workflow Pane**  . The Workflow pane pops out or hides.

2. Click on **Edit Pane**  . The Status Bar pops up or hides.

3. Click on **Layout Error Check**  . The layout error messages, if any, are shown or hidden.



Chapter

3

IBIS / RLGC Module: Net-Based Simulation Single-Die Single-BGA Packages

This chapter takes you through the steps to use the XtractIM tool IBIS / RLGC Module in the simulation of a single-die, single-BGA package.

Prepare for the Simulation

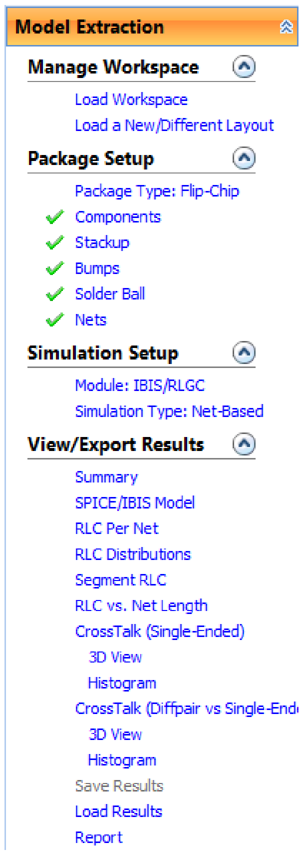
Collect this information before you begin the simulation.

- Have the Bump diameters, length, heights and conductivity ready
- Have the Stackup information ready
- Make sure your files have been translated into SPD format

Lesson One: Simulation Setup

Overview

Follow these steps to perform a typical workflow in Interaction Mode.



1. Load an existing workspace file (.xml file).
2. Load a layout file (.spd file).
3. Select a package type. Choose from:
   - Flip-chip
   - Single BGA (Board Grid Array)
   - Stacked BGA
   - Wirebond
4. Setup the components. You can select or deselect **Die component** and **Board component**.
5. Setup the Stackup.
6. Set parameters for the Bump or Solderball medium layer.
   - Set the Bump data if it is a flip-chip package.
   - Set the Solder Ball data.
7. Select the nets for extraction.

8. Setup extraction frequency and capacitance or inductance output control.

Load Workspace and Layout File

1. Launch XtractIM.

   The XtractIM main window opens. Two icons are available:

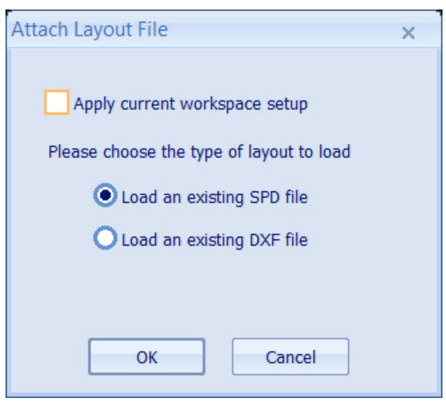
   New  and **Open** 

2. Click the **New** icon to create a new workspace or select:

   Workspace > New

3. Click **Load Workspace** in the **Workflow** pane and browse to load an existing workspace file.
4. To load the **Package Structure**, click **Load a New/Different Layout** in the **Workflow** pane.

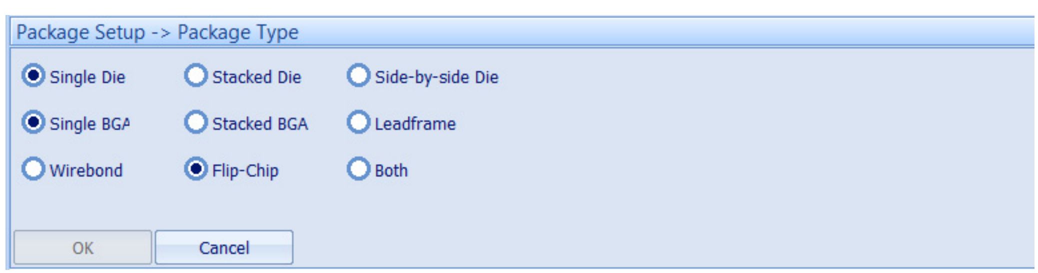
   The **Attach Layout File** window opens.



Select Package Type

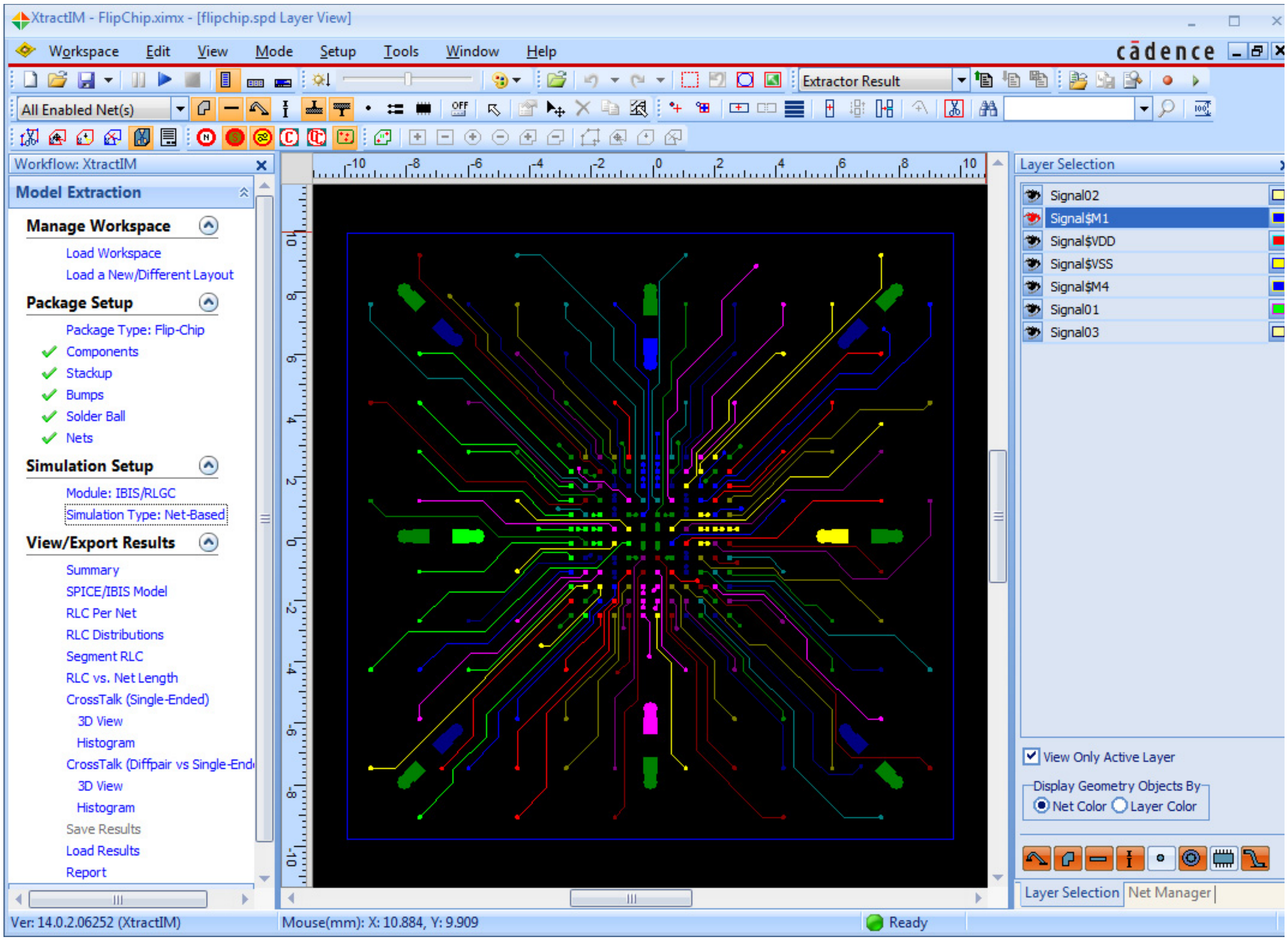
1. Click **Package Type: Flip-Chip** in the **Workflow** pane to select package type.

   The **Package Setup -> Package Type** pane appears at the bottom of the window.



2. Click **OK** to save your selection.

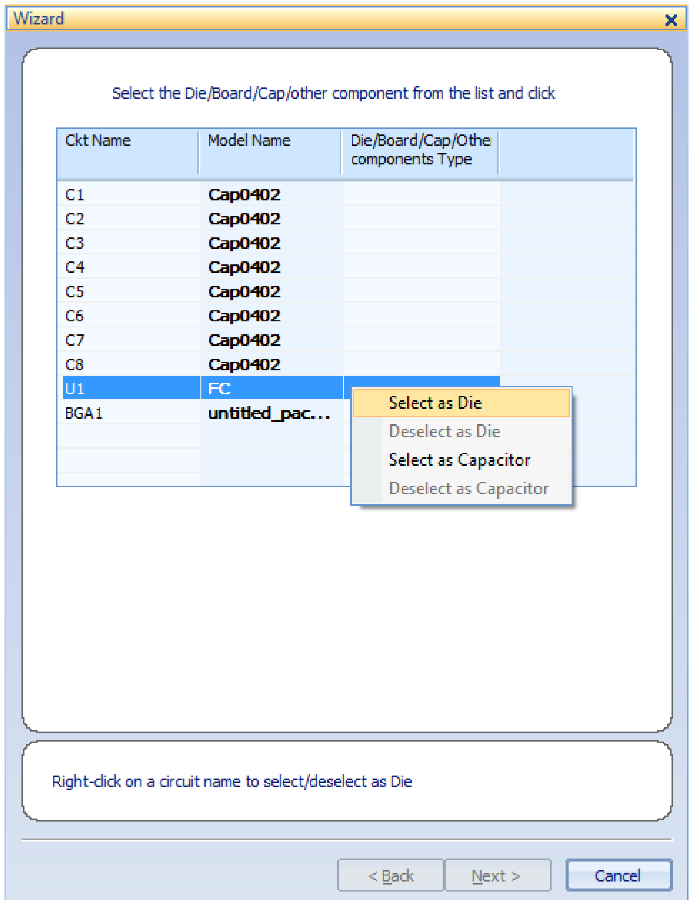
   The following example shows the workspace with the package type: **Single Die**, **Single BGA**, **Flip-Chip**.


XtractIM - FlipChip.ximx - [flipchip.spd Layer View]
Workspace Edit View Mode Setup Tools Window Help
cādence
Extractor Result
All Enabled Net(s)
Workflow: XtractIM
Model Extraction
Manage Workspace
Load Workspace
Load a New/Different Layout
Package Setup
Package Type: Flip-Chip
Components
Stackup
Bumps
Solder Ball
Nets
Simulation Setup
Module: IBIS/RLGC
Simulation Type: Net-Based
View/Export Results
Summary
SPICE/IBIS Model
RLC Per Net
RLC Distributions
Segment RLC
RLC vs. Net Length
CrossTalk (Single-Ended)
3D View
Histogram
Save Results
Load Results
Report
Layer Selection
Signal02
Signal$M1
Signal$VDD
Signal$VSS
Signal$M4
Signal01
Signal03
View Only Active Layer
Display Geometry Objects By
Net Color
Layer Color
Layer Selection
Net Manager
Ver: 14.0.2.06252 (XtractIM)
Mouse(mm): X: 10.884, Y: 9.909
Ready

Setup Components

1. Click **Components** in the **Workflow** pane to set up the Components data for a Flip-Chip package.

   The **Wizard** pane opens.



2. Right-click the desired component and select **Select as Die** from the pop-up menu list.
3. Click **Next**.
4. Right-click the desired component and select it as a **Board** component.
5. Click **Finish** to complete the components setup.

Setup Stackup

1. Click **Stackup** in the **Workflow** pane.

   The **Layer Manager -> Stack Up** window opens.
2. Right-click **Signal$Bottom** layer.
3. Insert a **Solder Ball Medium Layer**.
4. Insert an empty Signal layer.

5. Insert a medium layer standing for a PCB medium layer. All layers are inserted under **Signal$Bottom**.

   The added Signal layer is located at the end of the solder ball.

Setup Bump Medium Layer

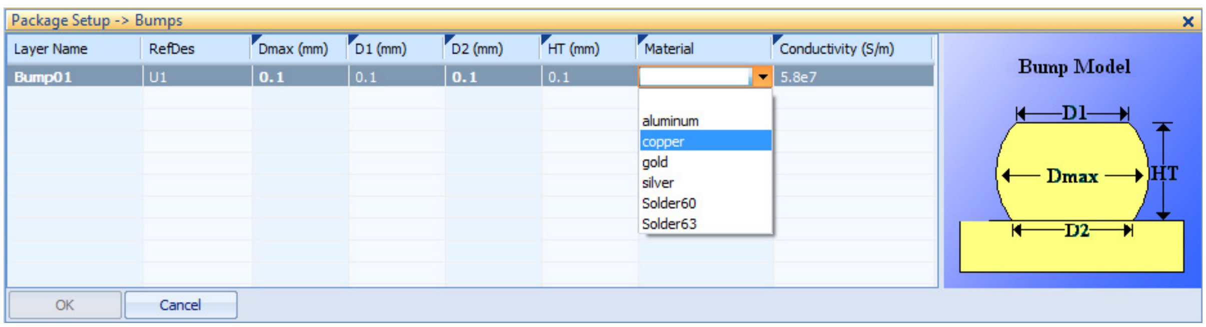
1. Right-click on the **Signal$Top** layer.
2. Insert a **Bump Medium Layer** above **Signal$Top**.
3. Insert a **signal layer** above Signal$Top. The added signal layer is the ending of the Bump.

   Both the Bump and Solder Ball Medium Layers are created.

Setup Bumps

1. Click **Bumps** in the **Workflow** pane to set up the Bump data for a Flip-Chip package.

   The **Package Setup -> Bumps** pane appears at the bottom of the window.



2. Input the settings for the Bumps.

   Maximum Diameter: Dmax (mm)

   D1 (mm)

   D2 (mm)

   Height: HT (mm)

   Conductivity (S/m) (or click **Material** and choose material from the drop-down list, the conductivity will be automatically input.)

NOTE!	XtractIM includes a material file in library (in the default intallation path: <INSTALL_DIR>\SpeedXP\Library\material\). It can be edited.

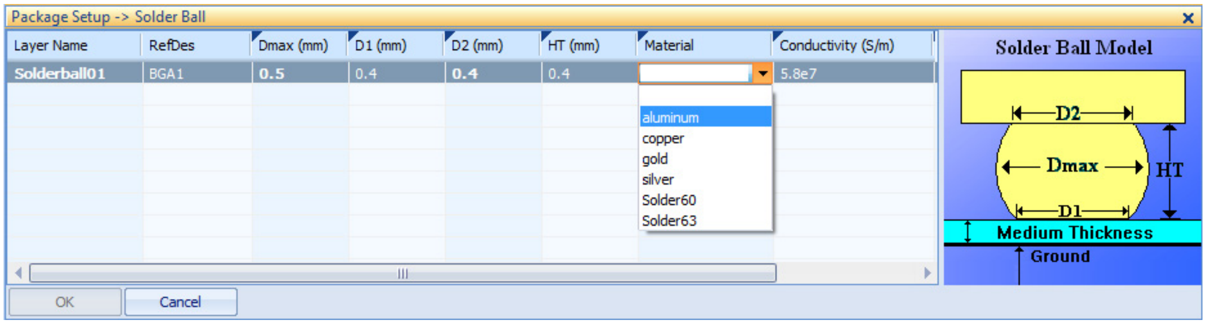
3. Click **OK** to save your entries.
4. (Optional) Click **Cancel** if you do not want to save your changes, or if you want to start over and re-enter your settings.

NOTE!	Perform the same steps to set up a wirebond package.

Setup Solder Ball

1. Click **Solder Ball** in the **Workflow** pane to setup the Solder Ball data for a Flip-Chip package.

   The **Package Setup -> Solder Ball** pane appears at the bottom of the window.



2. Input the settings for Solder Ball.

   Maximum Diameter: Dmax (mm)

   D1 (mm)

   D2 (mm)

   Height: HT (mm)

   Conductivity (S/m) (or click **Material** and choose material from the drop-down list, the conductivity will be automatically input.)

3. Click **OK** to save your entries.

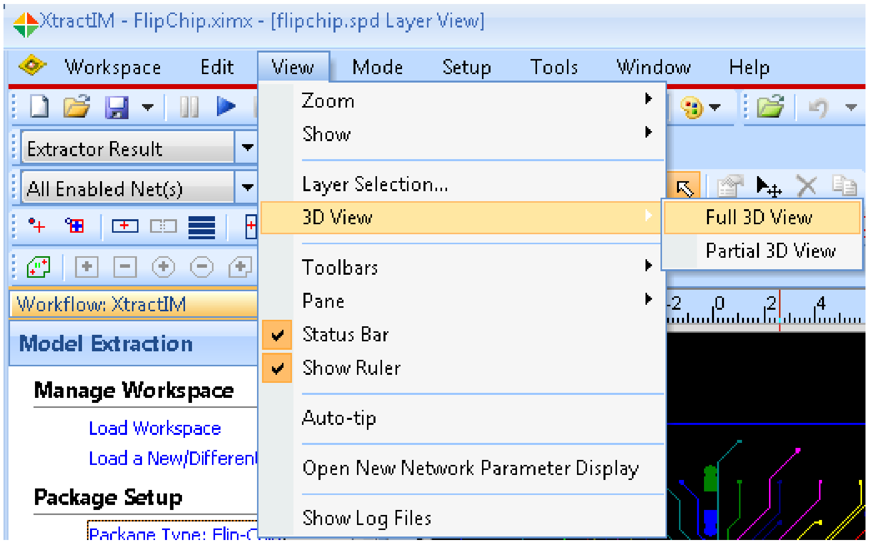
4. (Optional) Click **Cancel** if you do not want to save your changes, or if you want to start over and re-enter your settings.

Viewing 3D Layout

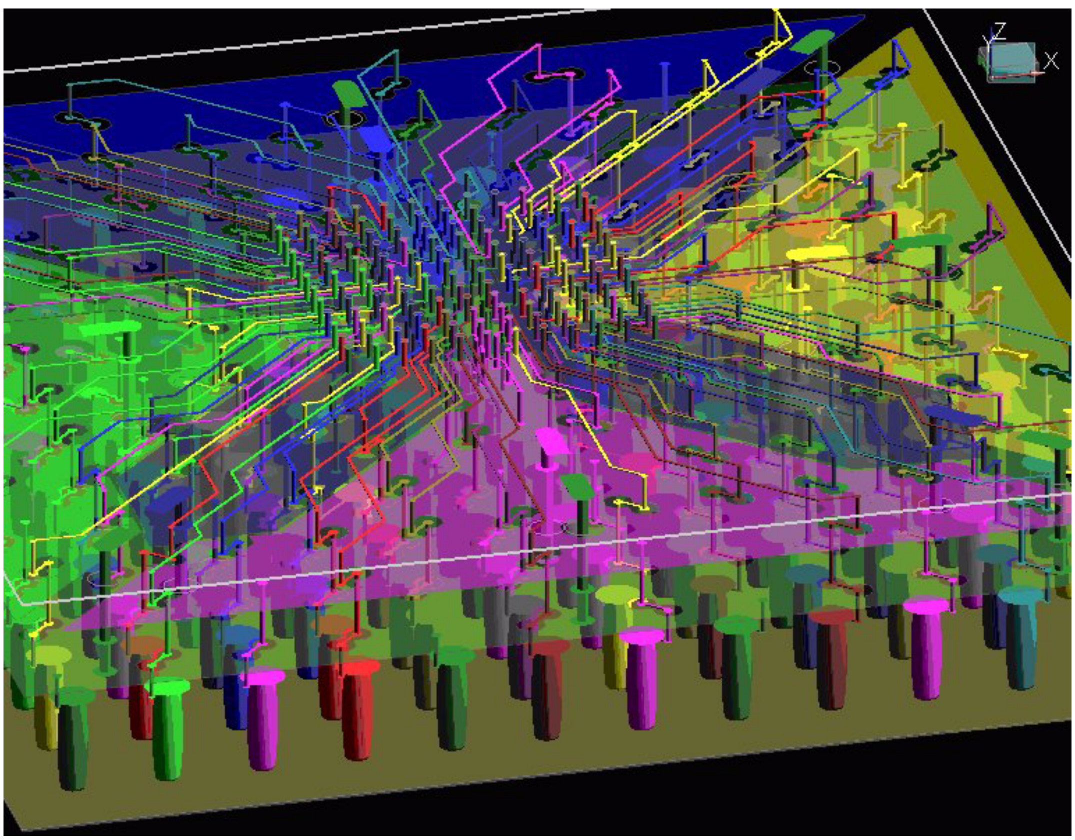
Full 3D View

Select

View > 3D View > Full 3D View



The 3D view of the package is displayed showing the bumps and solder balls.

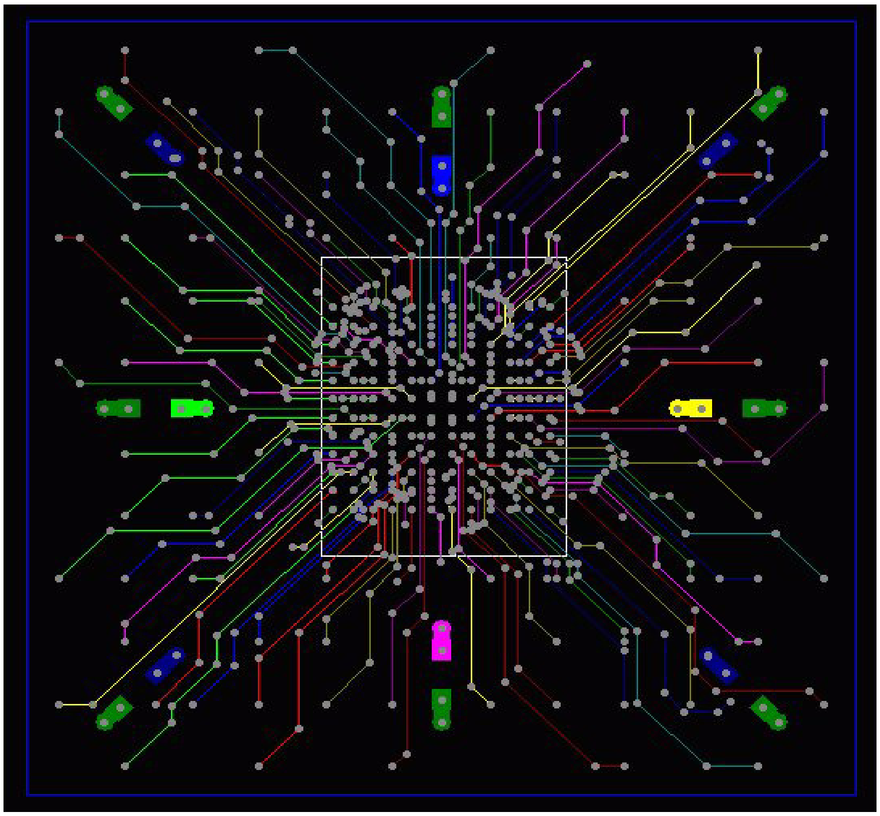


Partial 3D View

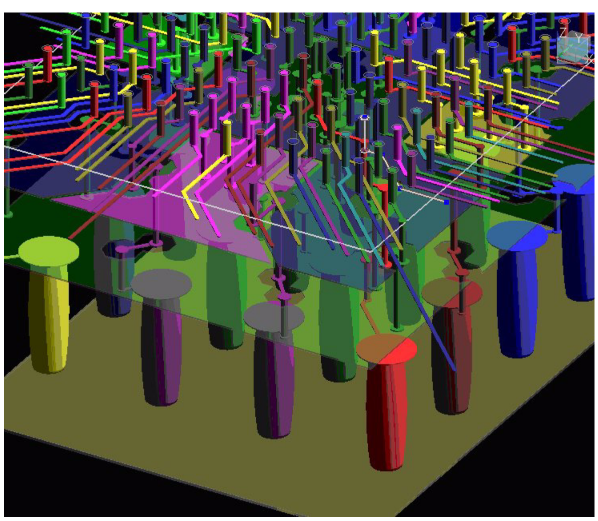
1. Select

   View > 3D View > Partial 3D View

2. Use the mouse to select a region (shown as a box outline in the following image).



The 3D view of the selected region is shown in the display.

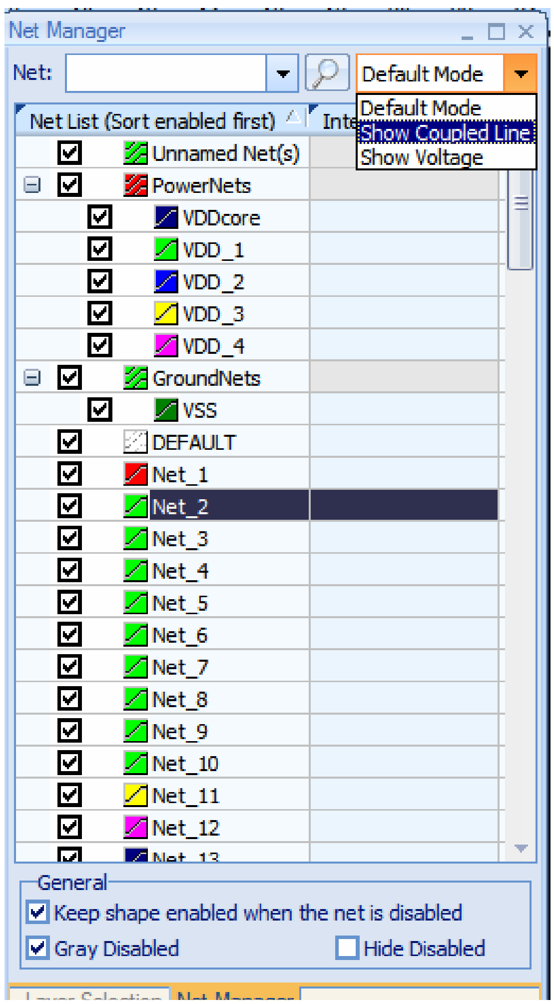


Setup Nets

1. Click **Nets** in the **Workflow** pane to open **Net Manager**.
2. Choose any desired nets for **RLC** (Resistance, Inductance and Capacitance)extraction.

   You can move the Signal Net into and out of Power Nets and Ground Nets if you wish.
3. Choose only the desired Ground Net as the reference net.

   At least one Ground Net must be selected to act as a **Reference ground Net**.
4. Set the Trace coupling threshold to a proper level to generate RLC models with accurate mutual terms.
5. Switch to **Show Coupled Line**. The **Coupled Lines Edit** pane opens.
6. Set the **Trace Coupling Threshold** to a proper level.



NOTE!	The Coupled Lines tab controls only the coupling threshold between Traces. Via coupling and wirebond coupling are always considered.

Rise Time and %Coupling

When Traces are identified as **coupled lines**, the crosstalk between these lines is calculated during the simulation. Based on electrical threshold parameters, Traces belonging to several nets are automatically identified and analyzed as coupled transmission line sections.

- Two Traces are said to be coupled if their reverse crosstalk exceeds the **%Coupling**
- Coupled lines can be selected after the relevant Traces have been placed. Coupled lines are treated as multi-conductor transmission lines
- An **isolated Trace** is modeled by the single transmission line algorithm

For a given **Rise Time**, the accumulated coupled section lengths between Trace nets should be long enough for forward crosstalk to exceed the %Coupling.

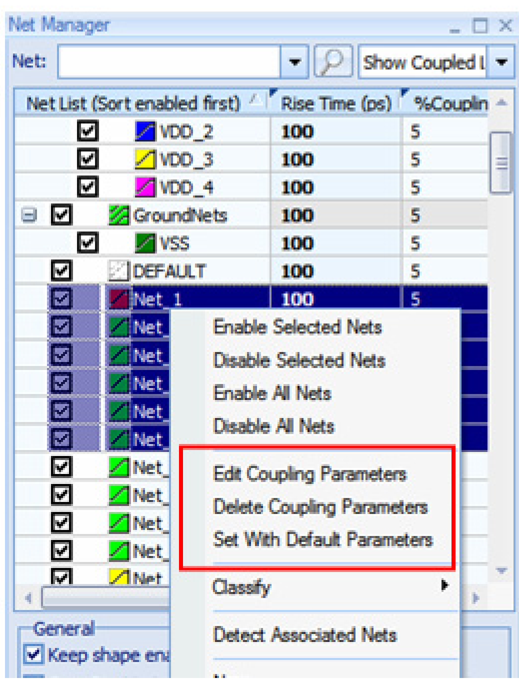
- **Rise Time Value** — Must be greater than 0. Default value is 200 ps
- **%Coupling Value** — Must be 0 < value ≤ 100. Default value is 5%

NOTE!	If the coupling parameters - %Coupling and Rise Time - are left blank, the trace-to-trace coupling is not calculated during simulation.

Setup Rise Time and %Coupling

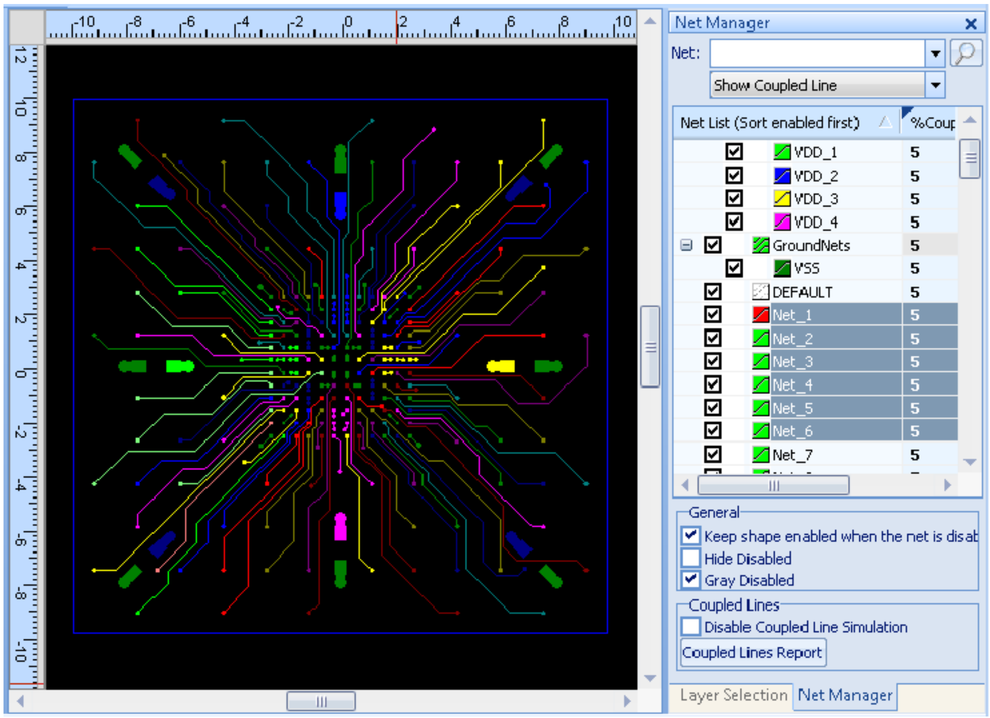
1. Click on **Coupled Line** tab to set up ***Setup Extraction Frequency*** to identify a coupled trace.
2. Select the nets you wish to edit.
3. Right-click to open the pop-menu.
4. Select:

   Set With Default Parameters



5. Select **Show Coupled Lines** in the **Net Manager**.

   The Trace identified as coupled lines is displayed in the **Layout View** pane, as shown in the following screen.

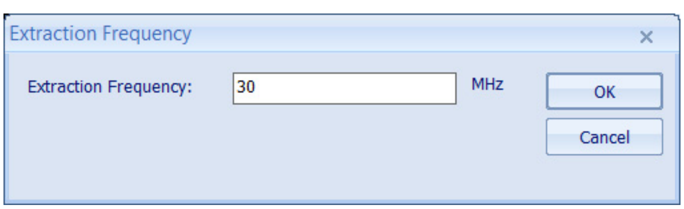


Setup Extraction Frequency

1. To change the Extraction Frequency, select

   Setup > Extraction Frequency

   The **Extraction Frequency** window opens.



2. Input the value of extraction frequency in the field.

   The default value is 30MHz.

3. Click **OK**.

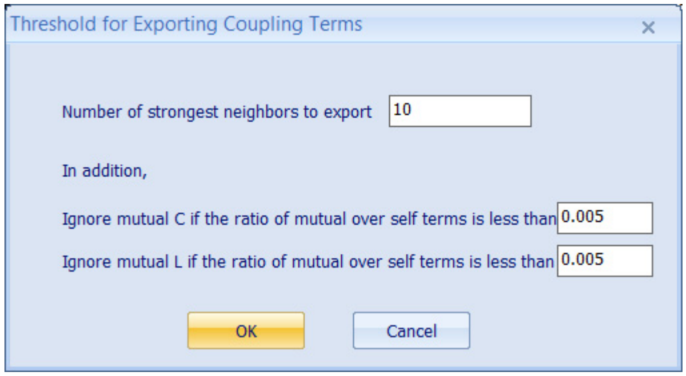
Setup Threshold for Exporting Mutual Terms

XtractIM provides an option to reduce the size of the output circuit during the export stage. Besides, XtractIM captures all the coupling during the extraction stage.

1. Select

Setup > Threshold for Exporting Coupling Terms

The **Threshold for Exporting Coupling Terms** window opens.



2. Input the number of strongest coupling neighbors to be kept in the circuit model.

   The default number of strongest coupling neighbors is 10; which means outputting the 10 strongest neighbors (including self).

3. Ignore mutual capacitance or inductance if the ratio of mutual terms over self term is less than a percentage.

   The default percentage threshold for ignoring mutual capacitance and inductance is 0.005.

Exporting Mutual Terms Example

If the mutual capacitance or inductance is less than the 0.5% of the minimum of the two self-capacitances and inductances,

then

XtractIM does not output the mutual capacitance or inductance

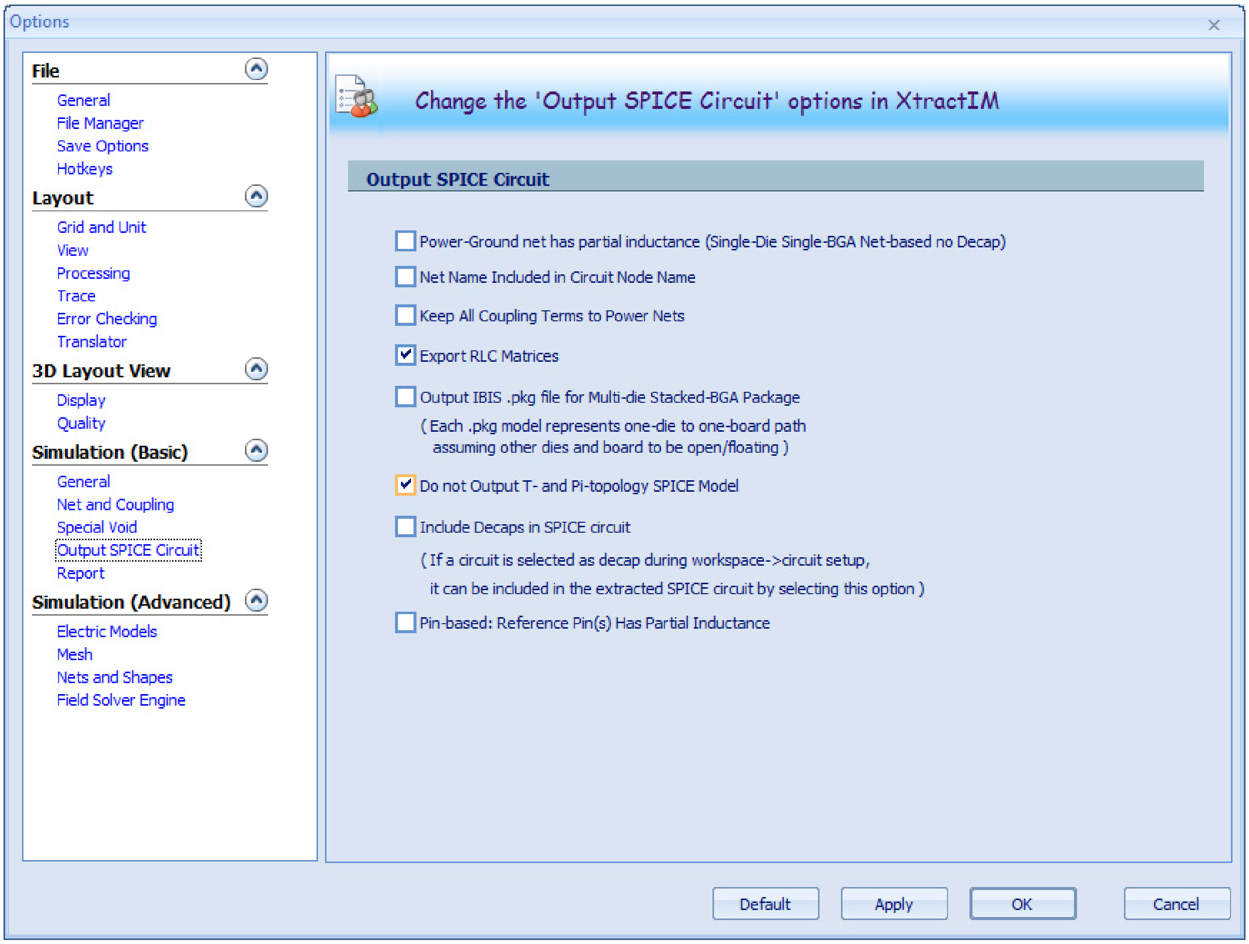
NOTE!	Threshold for Exporting Coupling Terms is used to: - Control the size of the SPICE circuit file. - Ignore mutual capacitance or inductance less than a specified net-to-net coupling threshold.

Output RLC Matrix in .csv Format

1. To output RLC matrix in .csv format, select

   Tools > Options > Edit Options...

   The **Options** window opens.

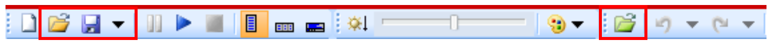


2. Click **Output SPICE Circuit** under **Simulation (Basic)**.
3. Select the **Export LC Matrices** option.
4. (Optional) Select the **Do not Output T- and Pi-topology SPICE Model** option if you do not want these two SPICE models.
5. Click **OK**.

Lesson Two: Save Work Areas

Tool Bar

There are four icons related with work areas saving.



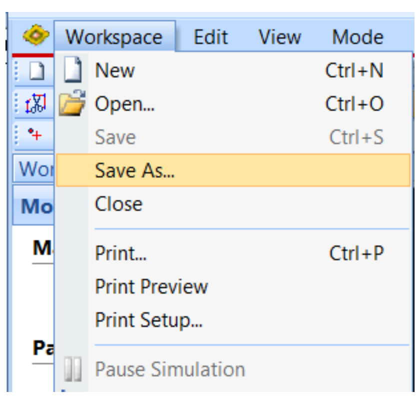
Icon	Description
	Open Workspace File
	Save all
	Save Workspace File Save Layout File
	Attach a layout file

NOTE!	Saving the workspace automatically saves the .spd file. Saving the layout file does not automatically save the workspace.

Workspace File

To save the workspace under a different name, select

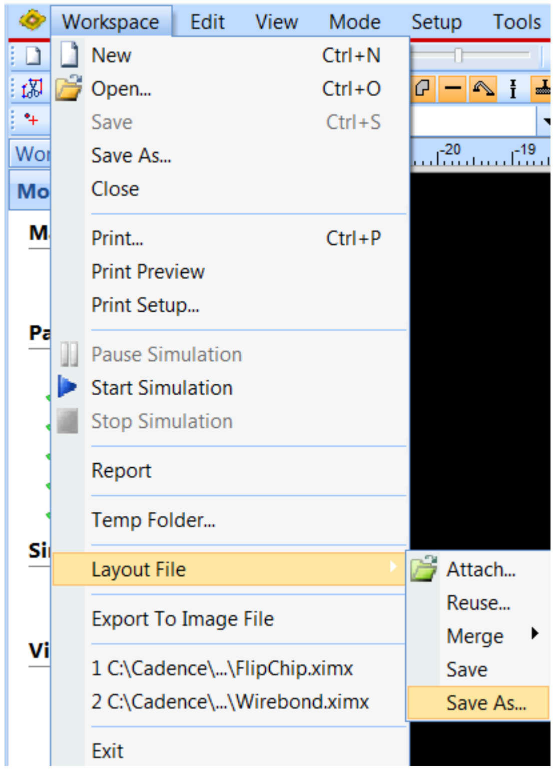
Workspace > Save As...



Layout File

To save the layout file under a different name, select

Workspace > Layout File > Save As...


Workspace
Edit
View
Mode
Setup
Tools
New Ctrl+N
Open... Ctrl+O
Save Ctrl+S
Save As...
Close
Print... Ctrl+P
Print Preview
Print Setup...
Pause Simulation
Start Simulation
Stop Simulation
Report
Temp Folder...
Layout File
Export To Image File
1 C:\Cadence\...\FlipChip.ximx
2 C:\Cadence\...\Wirebond.ximx
Exit
Attach...
Reuse...
Merge
Save
Save As...

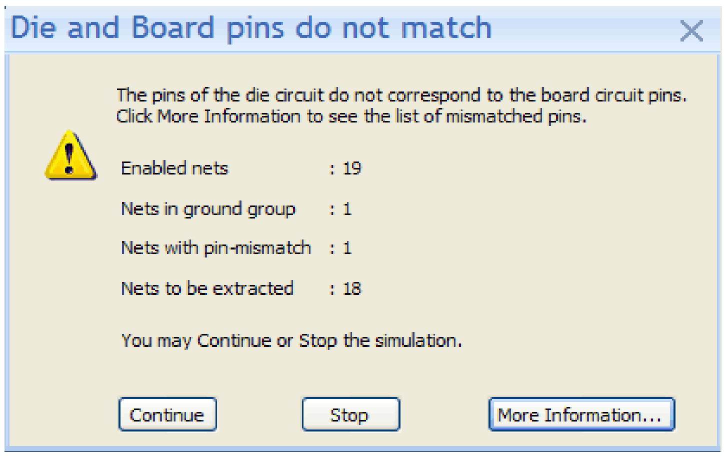
LESSON THREE: RUN THE SIMULATION

Start Simulation

1. Click the **Start Simulation** icon  on toolbar to start the extraction (simulation).

   XtractIM only extracts RLCG for a net which has at least one pin at the Die side and at least one pin at the board side.

   At the beginning of the simulation, if some nets have Die-Board mis-match, a pop-up window appears.

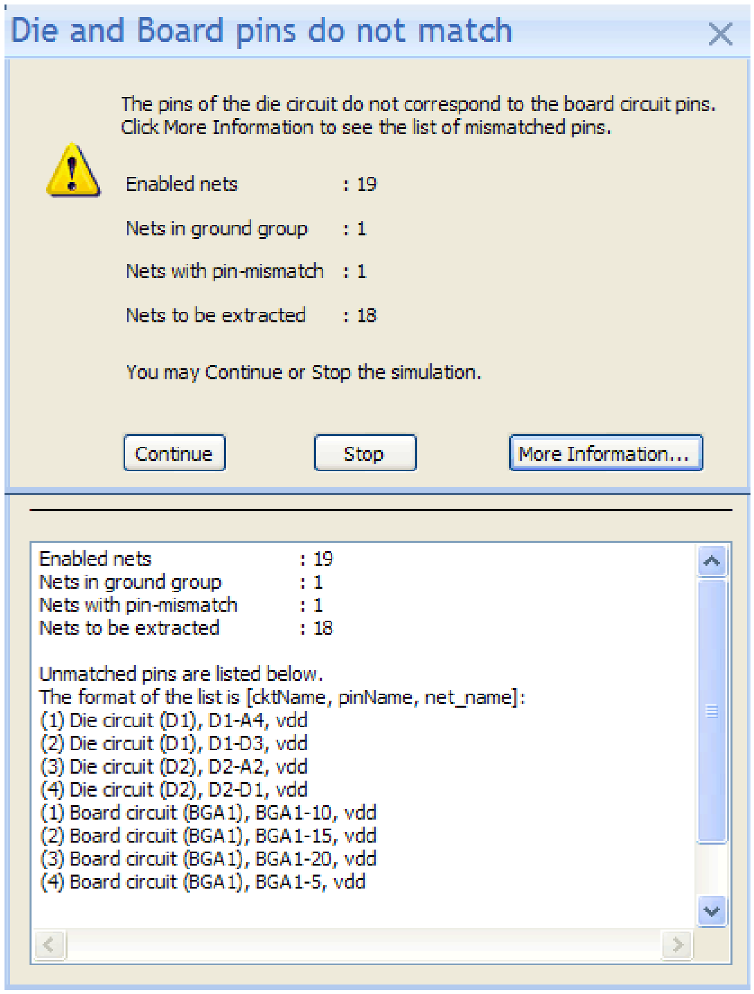


2. Select the next action.
   - **Continue** — Continue the simulation
   - **More Information** — Examine what nets are mis-matched
   - **Stop** — Cancel the simulation

Investigate Mis-matched Nets

1. The **More Information** window lists all the mis-matched nets.
2. Investigate the mis-matched nets to see whether there is a special design or a defective design.
3. Decide whether or not to proceed with the simulation.
4. Choose **Continue**, **Stop** or **More Information**.

   If 30 seconds pass and the user has not made a choice; then, by default, the simulation continues.



NOTE!	A file named Xtracted_PinNode_Info.log is saved on hard disk. This file records the total extracted node number of each circuit and, if available, the mis-matched information.

Lesson Four: Observe and Save Simulation Results

XtractIM performs calculations for each net. The calculations include:

- Conductance
- Mutual capacitance with other net
- Mutual loop inductance
- Resistance
- Self capacitance
- Self loop inductance

The inductance and capacitance are matrices. In the Inductance Matrix, the Diagonal Element is the Self Inductance of each net. The off-diagonal elements are mutual Inductance.

There are two concepts of capacitance matrix:

- Maxwell capacitance matrix
- SPICE capacitance matrix

- **Maxwell Capacitance Matrix** — Each diagonal element is the loading capacitance; for example, capacitance-to-ground when other nets are grounded. This represents the worst-case capacitive loading. Off-diagonal elements are mutual capacitance with negative values
- **SPICE Capacitance Matrix** — Each diagonal element is capacitance-to-ground. Off-diagonal elements are mutual capacitance with positive values

In the examples, review the relationship between Maxwell capacitance and SPICE capacitance matrix.

Inductance and Capacitance Matrix Example

A simple group of four nets has the following inductance and capacitance matrix.

$$\begin{bmatrix} L_{11} & L_{12} & L_{13} & L_{14} \\ L_{21} & L_{22} & L_{23} & L_{24} \\ L_{31} & L_{32} & L_{33} & L_{34} \\ L_{41} & L_{42} & L_{43} & L_{44} \end{bmatrix} \qquad \begin{bmatrix} C_{11} & C_{12} & C_{13} & C_{14} \\ C_{21} & C_{22} & C_{23} & C_{24} \\ C_{31} & C_{32} & C_{33} & C_{34} \\ C_{41} & C_{42} & C_{43} & C_{44} \end{bmatrix}$$

Capacitance Example

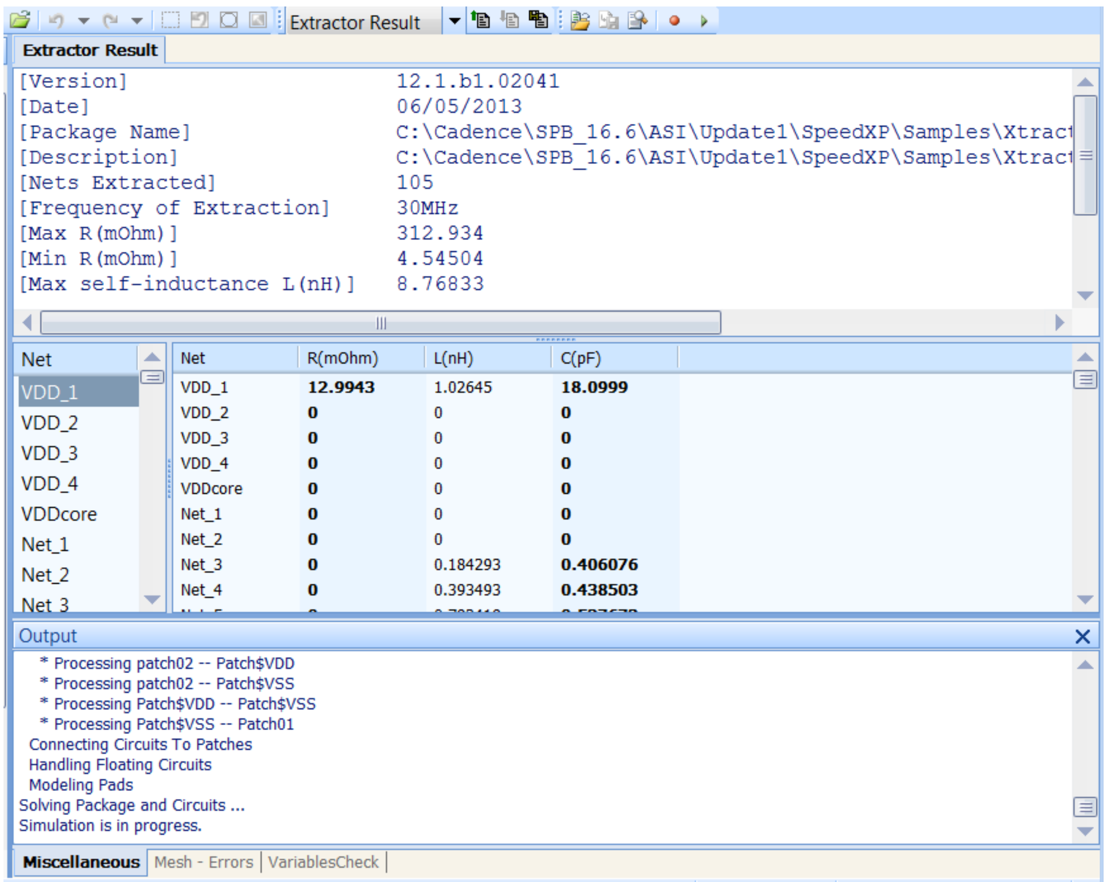
$$C_{ij}(SPICE) = -C_{ij}(Maxwell)$$

$$C_{ii}(Maxwell) = \sum_j C_{ij}(SPICE)$$

$$C_{ii}(SPICE) = \sum_j C_{ij}(Maxwell)$$

Summary of the Extracted Results

1. Click **Summary** in the **Workflow** pane.
2. View the data for R, L and C.
   - Extraction Frequency
   - Full RLC Matrix
   - Maximum / minimum R, L, C
   - Nets Extracted
   - Package Name
3. Reorder the list, if desired.



Save SPICE/IBIS Model Files

Upon completing the simulation, both model files are saved in the same directory as the **.spd** file.

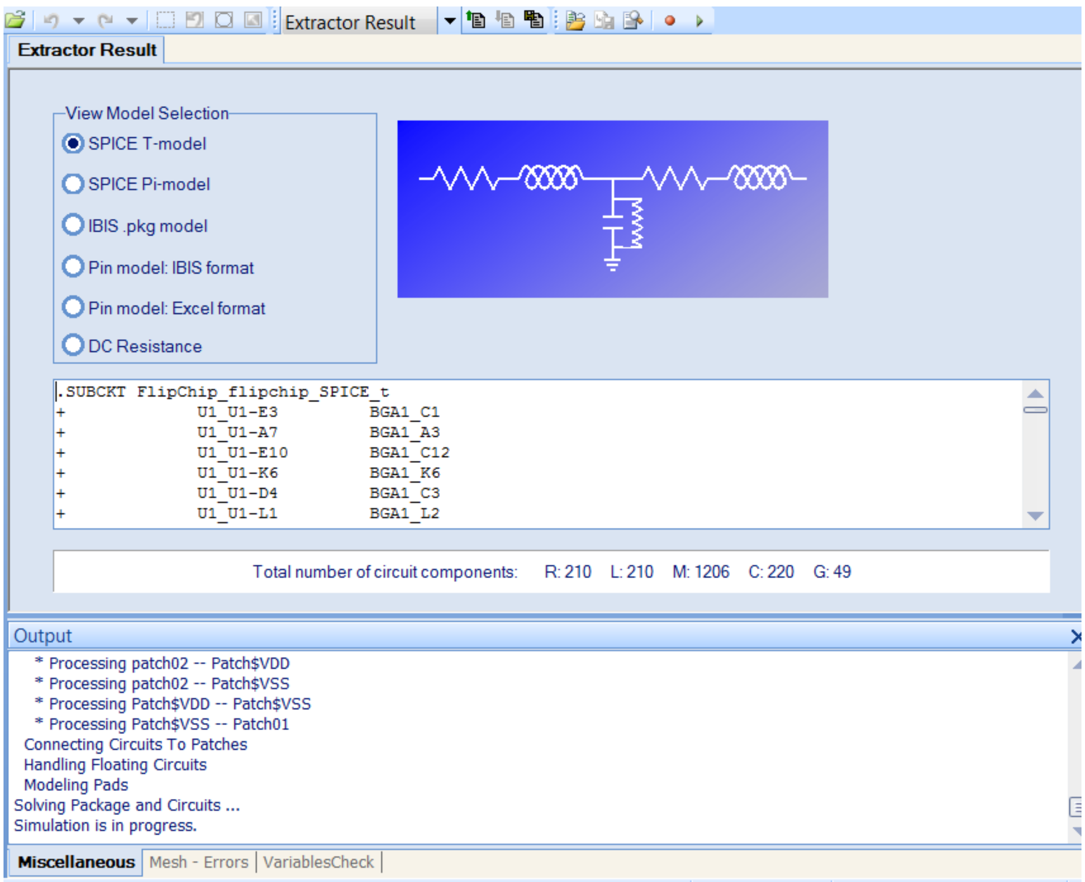
Save SPICE Model Files

The total number of elements in the circuit is displayed at the bottom of the window.

R, L, M, C and G where M is the mutual inductance

The SPICE Model is saved as a SPICE sub-circuit with the extension **.ckt**.

The SPICE model is a T-circuit.

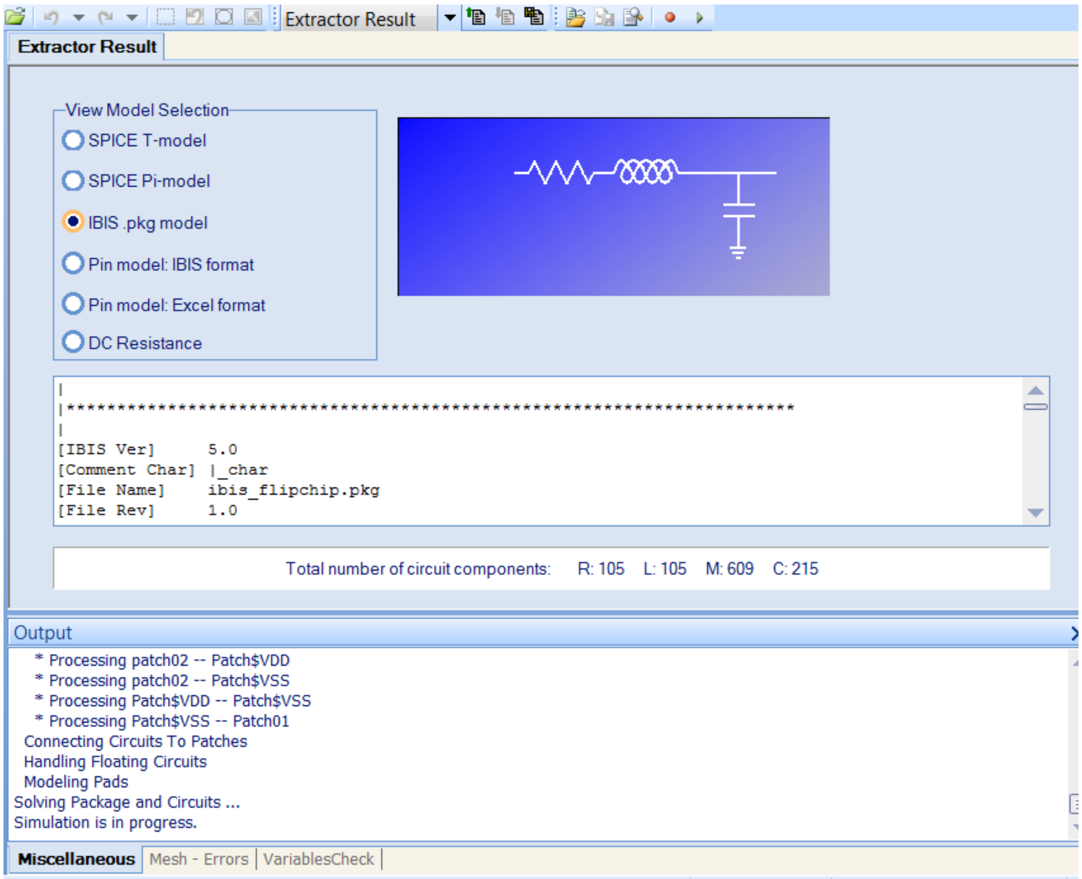


Save IBIS Model Files

The **IBIS Model** is saved as an IBIS package model. The saved model has the extension **.pkg**.

An .ibs format file is saved. The saved file includes each single net's R, L, C.

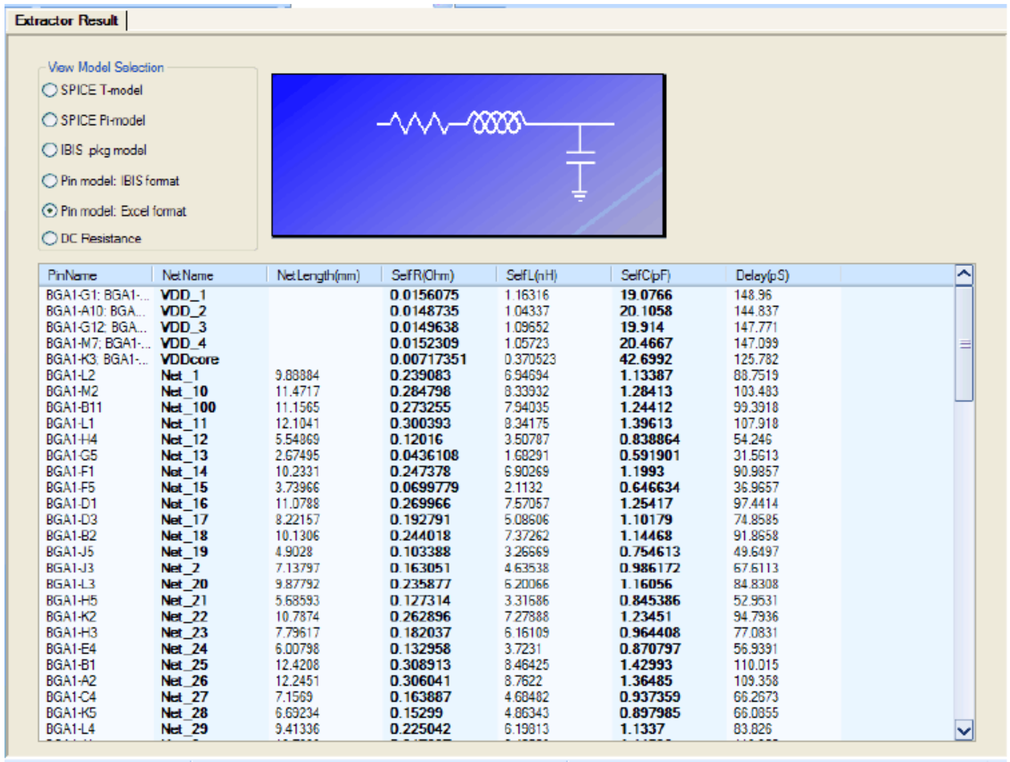
All power nets and ground nets are lumped together.



Load Pin Model in Excel Format

1. Click **Pin Model: Excel format** in the SPICE / IBIS Model window.
2. A .csv file is loaded. The .csv file includes information for each signal net.
   - Net Length
   - Net Name
   - Pin Name
   - Self-R
   - Self-L
   - Self-C
   - Time Delay

Self-C is the Maxwell Capacitance. The information for power nets does not include net length.

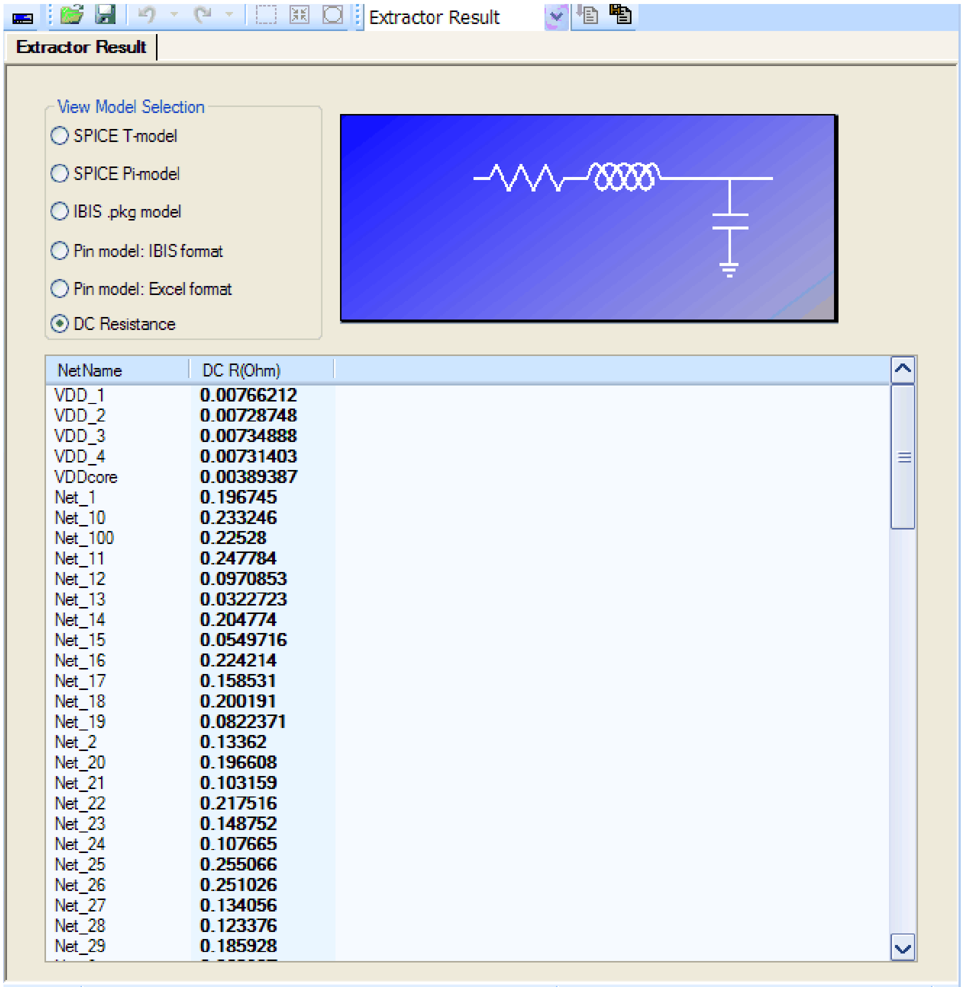


PinName	NetName	NetLength(mm)	SelfR(Ohm)	SelfL(nH)	SelfC(pF)	Delay(pS)
BGA1-G1; BGA1-...	VDD_1		0.0156075	1.16316	19.0766	148.96
BGA1-A10; BGA...	VDD_2		0.0148735	1.04337	20.1058	144.837
BGA1-G12; BGA...	VDD_3		0.0149638	1.09652	19.914	147.771
BGA1-M7; BGA1-...	VDD_4		0.0152309	1.05723	20.4667	147.099
BGA1-K3; BGA1-...	VDDcore		0.00717351	0.370523	42.6992	125.782
BGA1-L2	Net_1	9.88884	0.239083	6.94694	1.13387	88.7519
BGA1-M2	Net_10	11.4717	0.284798	8.33932	1.28413	103.483
BGA1-B11	Net_100	11.1565	0.273255	7.94035	1.24412	99.3918
BGA1-L1	Net_11	12.1041	0.300393	8.34175	1.39613	107.918
BGA1-H4	Net_12	5.54869	0.12016	3.50787	0.838864	54.246
BGA1-G5	Net_13	2.67495	0.0436108	1.68291	0.591901	31.5613
BGA1-F1	Net_14	10.2331	0.247378	6.90269	1.1993	90.9857
BGA1-F5	Net_15	3.73966	0.0699779	2.1132	0.646634	36.9657
BGA1-D1	Net_16	11.0788	0.269966	7.57057	1.25417	97.4414
BGA1-D3	Net_17	8.22157	0.192791	5.08606	1.10179	74.8585
BGA1-B2	Net_18	10.1306	0.244018	7.37262	1.14468	91.8658
BGA1-J5	Net_19	4.9028	0.103388	3.26669	0.754613	49.6497
BGA1-J3	Net_2	7.13797	0.163051	4.63538	0.986172	67.6113
BGA1-L3	Net_20	9.87792	0.235877	6.20066	1.16056	84.8308
BGA1-H5	Net_21	5.68593	0.127314	3.31686	0.845386	52.9531
BGA1-K2	Net_22	10.7874	0.262896	7.27888	1.23451	94.7936
BGA1-H3	Net_23	7.79617	0.182037	6.16109	0.964408	77.0831
BGA1-E4	Net_24	6.00798	0.132958	3.7231	0.870797	56.9391
BGA1-B1	Net_25	12.4208	0.308913	8.46425	1.42993	110.015
BGA1-A2	Net_26	12.2451	0.306041	8.7622	1.36485	109.358
BGA1-C4	Net_27	7.1569	0.163887	4.68482	0.937359	66.2673
BGA1-K5	Net_28	6.69234	0.15299	4.86343	0.897985	66.0855
BGA1-L4	Net_29	9.41336	0.225042	6.19813	1.1337	83.826

NOTE!	Net-length is reported only for single nets. If a single net has multi-pins in either die side or BGA side, its' net -length will NOT be reported.

View DC Resistance

1. Click **DC Resistance** in the SPICE / IBIS Model window.
2. View the **.csv** file. DC Resistance is given for each of the power, ground and signal nets. A **.csv** file is saved on the hard disk.

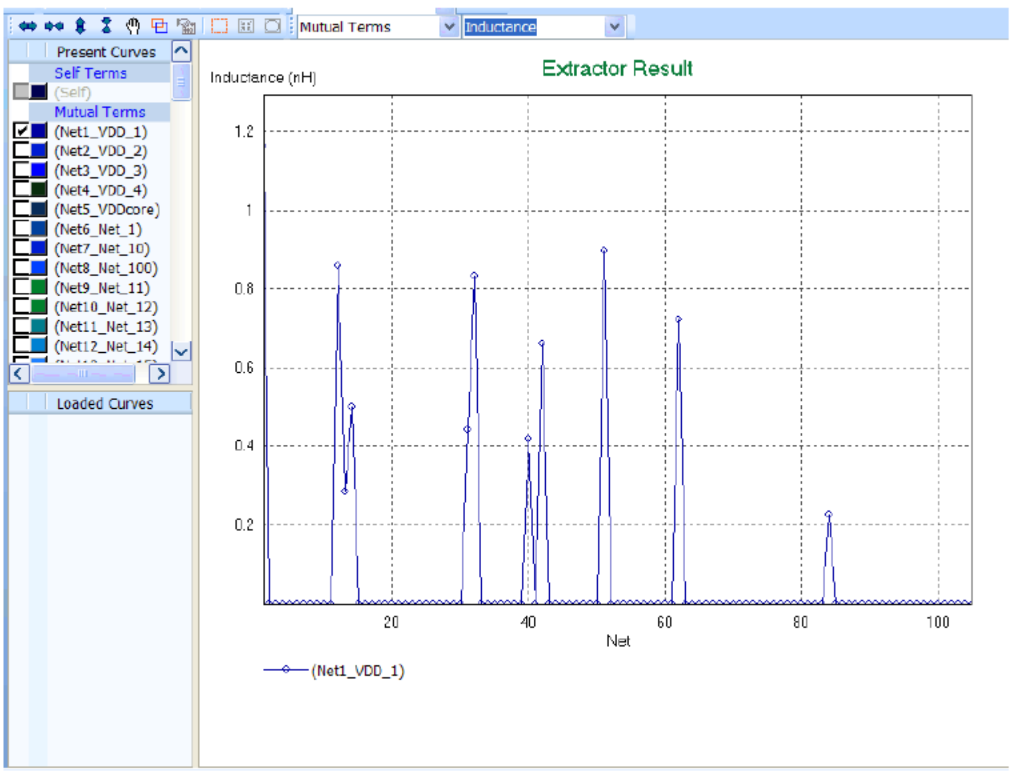


NetName	DC R(Ohm)
VDD_1	0.00766212
VDD_2	0.00728748
VDD_3	0.00734888
VDD_4	0.00731403
VDDcore	0.00389387
Net_1	0.196745
Net_10	0.233246
Net_100	0.22528
Net_11	0.247784
Net_12	0.0970853
Net_13	0.0322723
Net_14	0.204774
Net_15	0.0549716
Net_16	0.224214
Net_17	0.158531
Net_18	0.200191
Net_19	0.0822371
Net_2	0.13362
Net_20	0.196608
Net_21	0.103159
Net_22	0.217516
Net_23	0.148752
Net_24	0.107665
Net_25	0.255066
Net_26	0.251026
Net_27	0.134056
Net_28	0.123376
Net_29	0.185928

View RLC Per Net

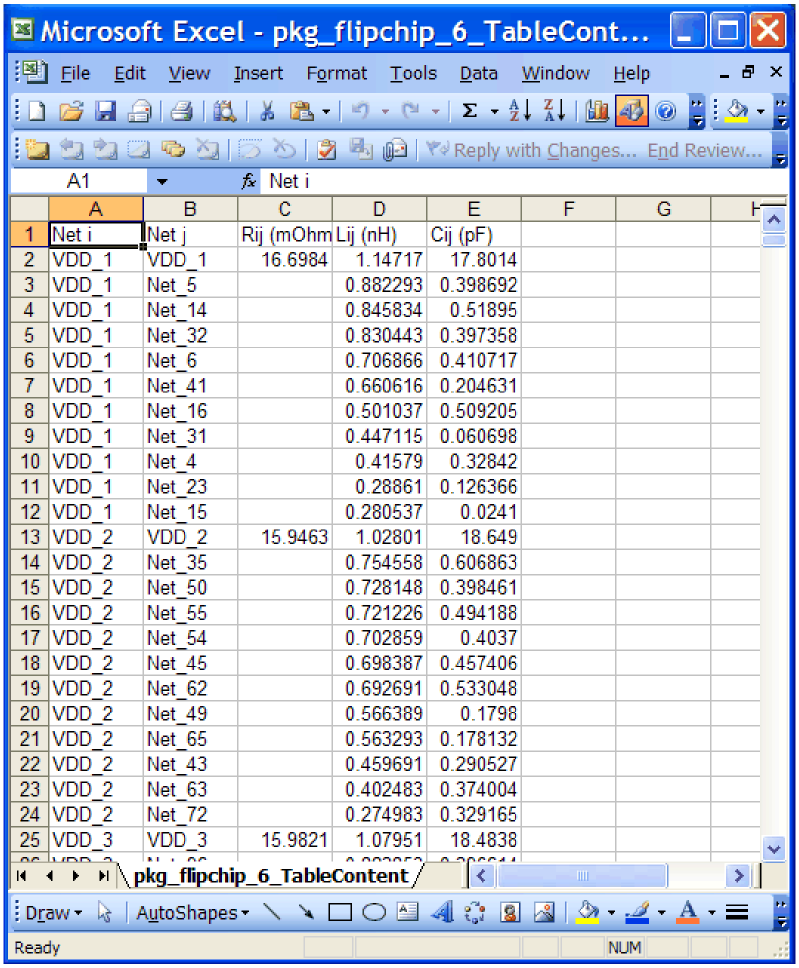
1. View **Resistance** for each net.
2. View **Self-inductance** for each net.
3. View **Self-capacitance** for each net.
4. View **Mutual Terms**.
5. View **Mutual Inductance**.
6. View **SPICE Mutual Capacitance**.

   Usually conductance is very low, so there is no view for conductance.



Save R / L / C Full Matrix

1. Click on **R/L/C** full matrix.
2. The matrix is saved on hard disk in .csv format.

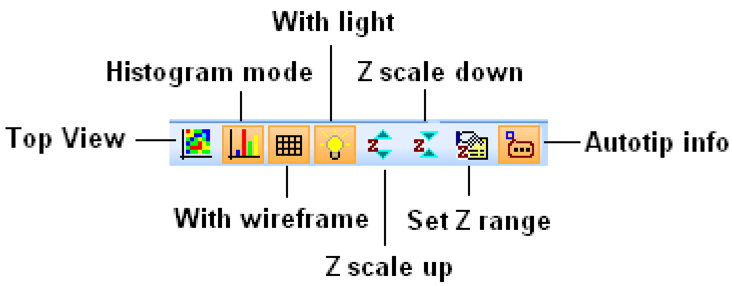


	A	B	C	D	E
1	Net i	Net j	Rij (mOhm	Lij (nH)	Cij (pF)
2	VDD_1	VDD_1	16.6984	1.14717	17.8014
3	VDD_1	Net_5		0.882293	0.398692
4	VDD_1	Net_14		0.845834	0.51895
5	VDD_1	Net_32		0.830443	0.397358
6	VDD_1	Net_6		0.706866	0.410717
7	VDD_1	Net_41		0.660616	0.204631
8	VDD_1	Net_16		0.501037	0.509205
9	VDD_1	Net_31		0.447115	0.060698
10	VDD_1	Net_4		0.41579	0.32842
11	VDD_1	Net_23		0.28861	0.126366
12	VDD_1	Net_15		0.280537	0.0241
13	VDD_2	VDD_2	15.9463	1.02801	18.649
14	VDD_2	Net_35		0.754558	0.606863
15	VDD_2	Net_50		0.728148	0.398461
16	VDD_2	Net_55		0.721226	0.494188
17	VDD_2	Net_54		0.702859	0.4037
18	VDD_2	Net_45		0.698387	0.457406
19	VDD_2	Net_62		0.692691	0.533048
20	VDD_2	Net_49		0.566389	0.1798
21	VDD_2	Net_65		0.563293	0.178132
22	VDD_2	Net_43		0.459691	0.290527
23	VDD_2	Net_63		0.402483	0.374004
24	VDD_2	Net_72		0.274983	0.329165
25	VDD_3	VDD_3	15.9821	1.07951	18.4838

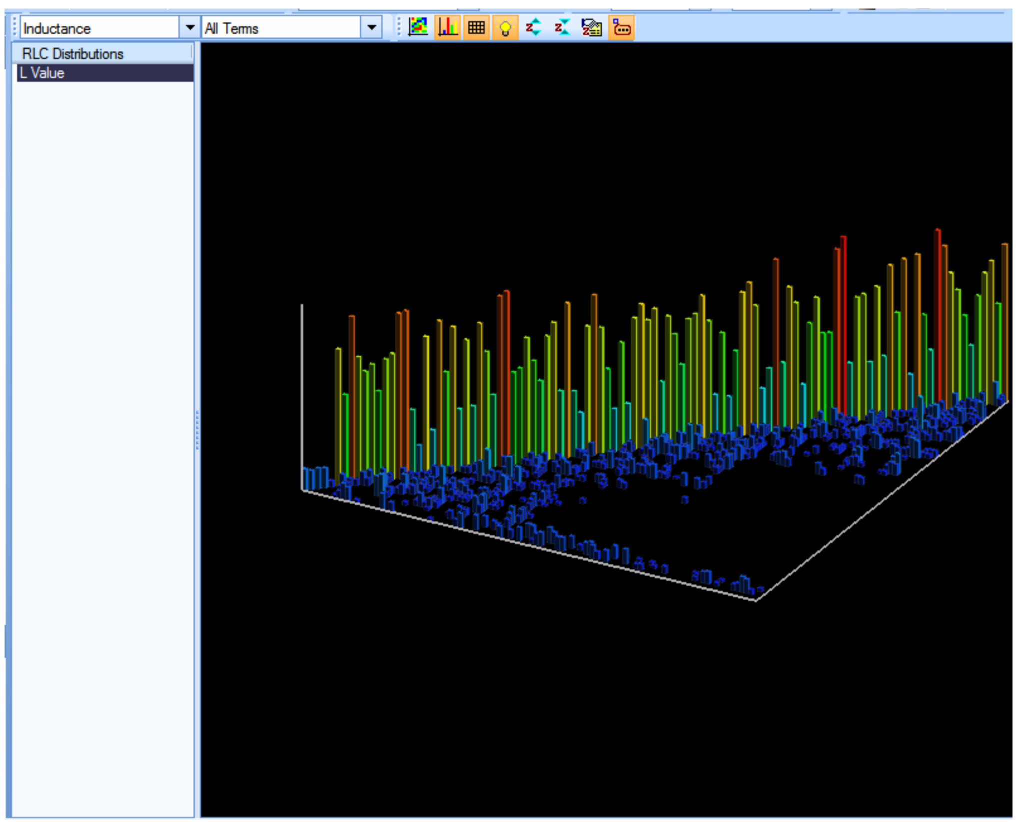
View RLC Distributions

1. Click **RLC Distributions** in the Workflow pane.
2. View the full matrix value of R, L and C.

   RLC Distributions offers eight kinds of views.



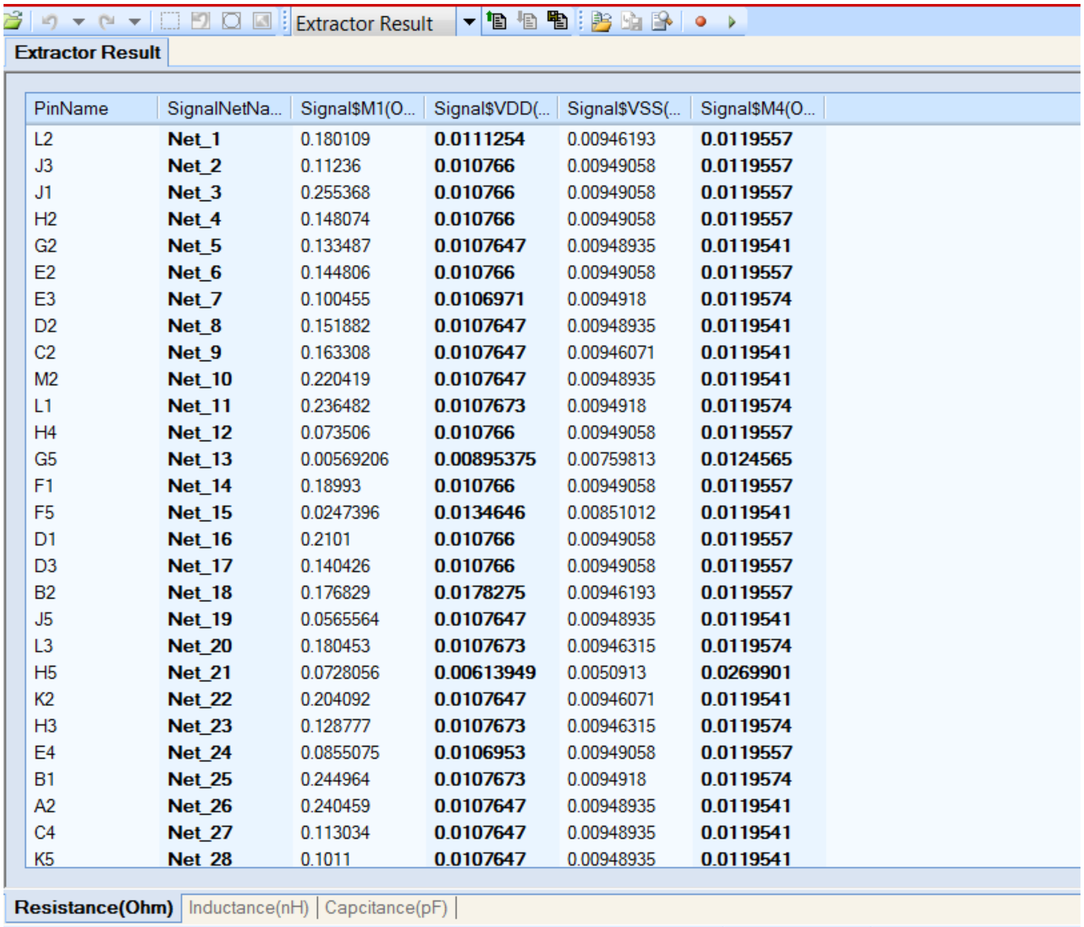
3. View the **RLC distributions**.



View Segment RLC

1. Click **Segment RLC** in the **Workflow** pane.

   For each Signal Net, XtractIM outputs the RLC segment of each Metal layer.



PinName	SignalNetNa...	Signal$M1(O...	Signal$VDD(...	Signal$VSS(...	Signal$M4(O...
L2	Net_1	0.180109	0.0111254	0.00946193	0.0119557
J3	Net_2	0.11236	0.010766	0.00949058	0.0119557
J1	Net_3	0.255368	0.010766	0.00949058	0.0119557
H2	Net_4	0.148074	0.010766	0.00949058	0.0119557
G2	Net_5	0.133487	0.0107647	0.00948935	0.0119541
E2	Net_6	0.144806	0.010766	0.00949058	0.0119557
E3	Net_7	0.100455	0.0106971	0.0094918	0.0119574
D2	Net_8	0.151882	0.0107647	0.00948935	0.0119541
C2	Net_9	0.163308	0.0107647	0.00946071	0.0119541
M2	Net_10	0.220419	0.0107647	0.00948935	0.0119541
L1	Net_11	0.236482	0.0107673	0.0094918	0.0119574
H4	Net_12	0.073506	0.010766	0.00949058	0.0119557
G5	Net_13	0.00569206	0.00895375	0.00759813	0.0124565
F1	Net_14	0.18993	0.010766	0.00949058	0.0119557
F5	Net_15	0.0247396	0.0134646	0.00851012	0.0119541
D1	Net_16	0.2101	0.010766	0.00949058	0.0119557
D3	Net_17	0.140426	0.010766	0.00949058	0.0119557
B2	Net_18	0.176829	0.0178275	0.00946193	0.0119557
J5	Net_19	0.0565564	0.0107647	0.00948935	0.0119541
L3	Net_20	0.180453	0.0107673	0.00946315	0.0119574
H5	Net_21	0.0728056	0.00613949	0.0050913	0.0269901
K2	Net_22	0.204092	0.0107647	0.00946071	0.0119541
H3	Net_23	0.128777	0.0107673	0.00946315	0.0119574
E4	Net_24	0.0855075	0.0106953	0.00949058	0.0119557
B1	Net_25	0.244964	0.0107673	0.0094918	0.0119574
A2	Net_26	0.240459	0.0107647	0.00948935	0.0119541
C4	Net_27	0.113034	0.0107647	0.00948935	0.0119541
K5	Net_28	0.1011	0.0107647	0.00948935	0.0119541

2. View the **Segment RLC values**.

Note the three tabs across the bottom of the screen. These bars are for Resistance, Inductance and Capacitance

- Click on the **Resistance** tab. The Resistance values are displayed.
- Click on the **Inductance** tab.The Inductance values are displayed.
- Click on the **Capacitance** tab. The Capacitance values are displayed.

Segment RLC Report

Segment RLC are reported for Signal Nets with single-pin at each of the Die- and Board circuit.

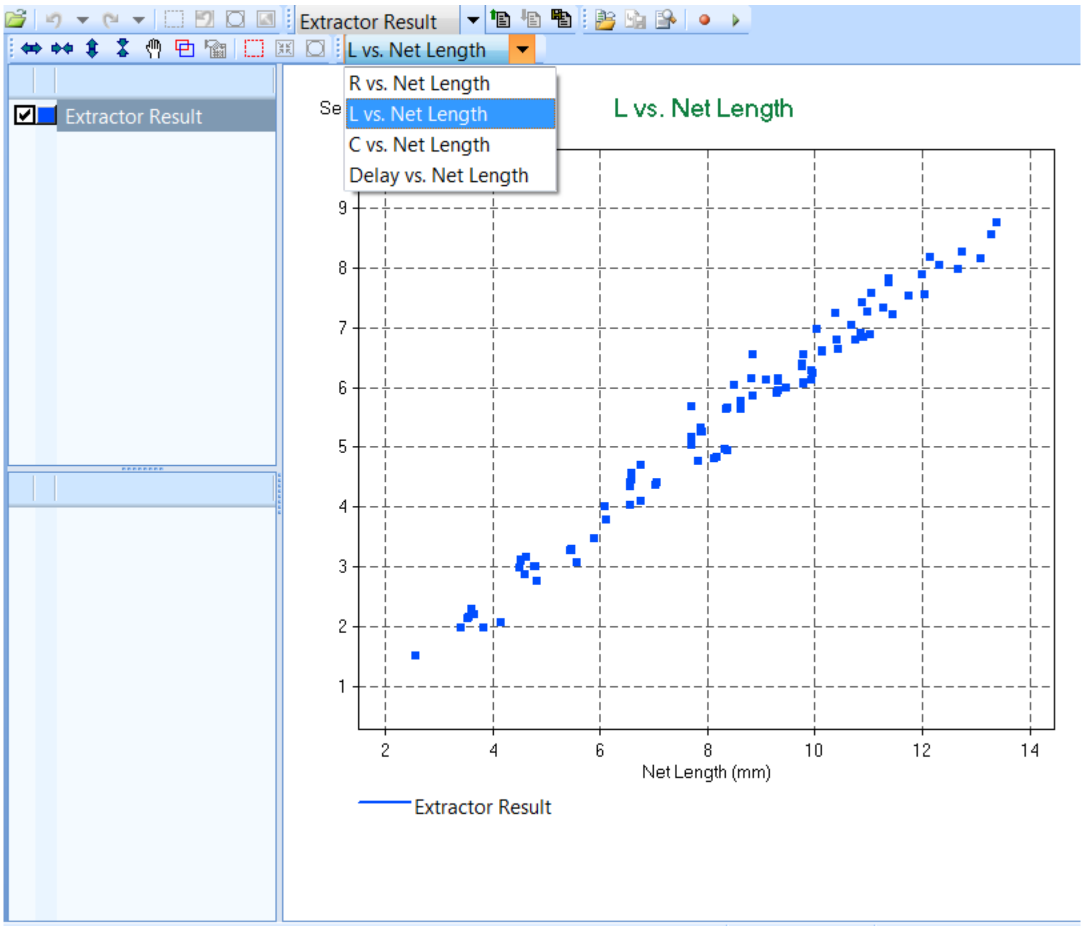
Via resistance and inductance are reported at the Via starting layer (the upper layer).

Components	Segment R	Segment L	Segment C
Wirebond	Yes	Yes	Yes
Trace	Yes	Yes	Yes
Via	Yes	Yes	Yes
Pad	-	-	Yes
Leadframe	Yes	Yes	Yes

View RLC vs. Net Length

1. Click **RLC vs. Net Length** in the **Workflow** pane.

   The **Inductance plot** is displayed in the result window.

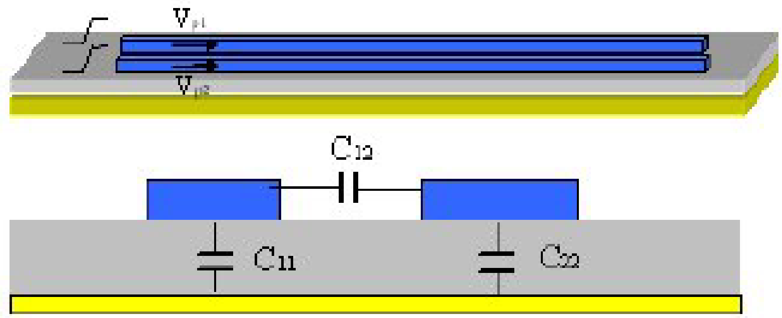


- To view the **Inductance plot**, select L vs. Net Length from the drop-down list
- To view the **Capacitance plot**, select C vs. Net Length from the drop-down list
- To view the **Delay plot, select**, select Delay vs. Net Length from the drop-down list

View Signal Nets Crosstalk

NEXT and FEXT

The mathematical definitions for NEXT and FEXT are shown in the following examples.



Aggressor: Z_{10}, L_{11}, C_{11}

Victim: Z_{20}, L_{22}, C_{22}

$$K = \frac{V_p}{4}\left(C_{12}Z_{20} + \frac{L_{12}}{Z_{10}}\right)$$

$$Where,\quad \frac{2}{V_p} = \frac{1}{V_{p1}} + \frac{1}{V_{p2}}$$

$$V_{p1} = \frac{1}{\sqrt{L_{11}C_{11}}} \qquad V_{p2} = \frac{1}{\sqrt{L_{22}C_{22}}}$$

$$Z_{10} = \sqrt{\frac{L_{11}}{C_{11}}} \qquad Z_{20} = \sqrt{\frac{L_{22}}{C_{22}}}$$

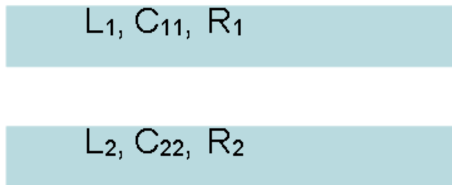
Definitions

K	The near-end coupling coefficient from aggressor net on victim net.
C_{11} / C_{22}	The self capacitance per unit length of signal net to its reference plane.
L_{11} / L_{22}	The self-inductance per unit length of signal net.
C_{12}	The mutual capacitance per unit length between aggressor and victim net.
L_{12}	The mutual inductance per unit length between aggressor and victim net.
V_{p1} / V_{p2}	The signal velocity propagate on signal net.
Z_{10} / Z_{20}	The characteristic impedance of signal Traces.

$$FEXT = \frac{\sqrt{L_{ii}C_{ii}}}{2T_r}\left(\frac{C_{ij}}{C_{ii}} - \frac{L_{ij}}{L_{ii}}\right)$$

RLC of differential Pair and crosstalk with others

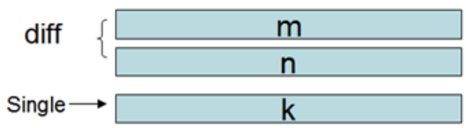
Assume two nets are different pair which has L1/ L2 and L12 as self-inductance and mutual inductance, and C11/C22 and C12 as loading capacitance and mutual capacitance



$$Diff\_R = R_1 + R_2$$
$$Diff\_L = L_1 + L_2 - 2L_{12}$$
$$Diff\_C = (C_{10} \cdot C_{20})/(C_{10} + C_{20}) + C_{12}$$
$$where,\ C_{10} = C_{11} - C_{12},\ C_{20} = C_{22} - C_{21}$$

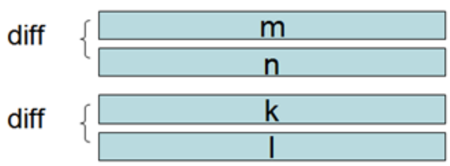
Mutual Inductance and Capacitance

- Differential Pair vs. Single-ended



$$L_{diff-k}:\quad L_{mn-k} = |L_{mk} - L_{nk}|$$
$$C_{diff-k}:\quad C_{mn-k} = C_{mk} + C_{nk}$$

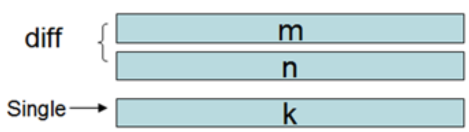
- Differential Pair vs. Differential Pair



$$L_{diff-diff}:\quad L_{mn-kl} = |(L_{mk} - L_{nk}) - (L_{ml} - L_{nl})|$$
$$C_{diff-diff}:\quad C_{mn-kl} = (C_{mk} + C_{nk}) + (C_{ml} + C_{nl})$$

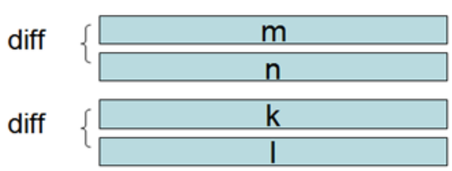
Crosstalk: NEXT and FEXT

- Differential Pair vs Single-ended



$$NEXT:\quad K^{near}_{mn-k} = |K^{near}_{m-k} - K^{near}_{n-k}|$$
$$FEXT:\quad K^{far}_{mn-k} = (K^{far}_{m-k} - K^{far}_{n-k})$$

- Differential Pair vs. Differential Pair

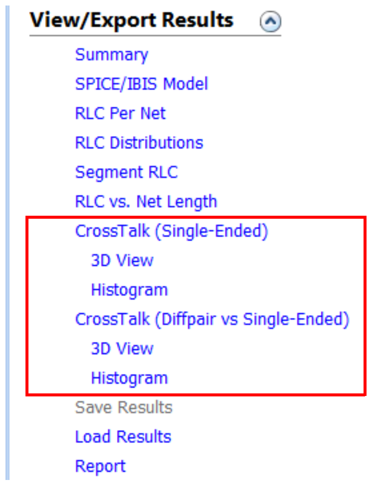


$$NEXT:\quad K^{near}_{mn-kl} = |(K^{near}_{m-k} - K^{near}_{n-k}) - (K^{near}_{m-l} - K^{near}_{n-l})|$$
$$FEXT:\quad K^{far}_{mn-k} = (K^{far}_{m-k} - K^{far}_{n-k}) - (K^{far}_{m-l} - K^{far}_{n-l})$$

View Crosstalk Results

Two kinds of Crossstalk results are provided:

- Single-Ended
- Diffpair vs Single-Ended



- To view single-ended near-end and far-end crosstalk (NEXT and FEXT) click **CrossTalk (Single-Ended)** in the **Workflow** pane
- To view difffpair near-end and far-end crosstalk (NEXT and FEXT) click **CrossTalk (Diffpair vs Single-Ended)** in the **Workflow** pane

A file is created in the project folder. The file name is:

*signal_Xtalk.csv

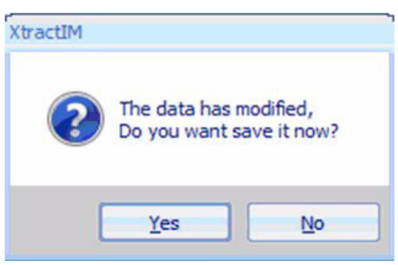
By default, the far-end crosstalk is defined with rise time = 100ps.

Net i	Net j	Rij (mOhm)	Lij (nH)	Cij (pF)	NEXT (%)	Total NEXT (...	FEXT (%)	Tr (pS)
Net_1	**Net_1**	220.586	**6.5703**	1.011		26.91		
Net_1	**Net_11**	0	**1.0701**	0.072	**5.33**		**-3.94**	100
Net_1	**Net_10**	0	**0.9105**	0.000	**3.14**		**-5.57**	100
Net_1	**Net_2**	0	**0.6407**	0.010	**3.43**		**-4.14**	100
Net_1	**Net_20**	0	**0.6277**	0.000	**2.48**		**-4.05**	100
Net_1	**Net_22**	0	**0.5013**	0.000	**1.85**		**-3.13**	100
Net_1	**Net_29**	0	**0.4415**	0.000	**1.80**		**-2.87**	100
Net_1	**Net_40**	0	**0.4049**	0.000	**1.58**		**-2.48**	100
Net_1	**Net_30**	0	**0.3650**	0.000	**1.73**		**-2.60**	100
Net_1	**Net_3**	0	**0.3007**	0.000	**1.05**		**-1.93**	100
Net_1	**Net_12**	0	**0.2742**	0.000	**1.56**		**-2.03**	100
Net_1	**Net_19**	0	**0.2512**	0.000	**1.45**		**-1.82**	100
Net_1	**Net_21**	0	**0.2459**	0.000	**1.52**		**-2.01**	100
Net_2	**Net_2**	143.402	**4.3835**	0.864		29.23		
Net_2	**Net_11**	0	**0.9447**	0.119	**6.30**		**-1.31**	100
Net_2	**Net_1**	0	**0.6407**	0.010	**3.03**		**-3.57**	100
Net_2	**Net_40**	0	**0.5993**	0.002	**2.74**		**-3.59**	100
Net_2	**Net_22**	0	**0.4411**	0.000	**1.84**		**-2.76**	100
Net_2	**Net_12**	0	**0.4346**	0.007	**3.14**		**-3.00**	100
Net_2	**Net_3**	0	**0.3957**	0.000	**1.55**		**-2.54**	100
Net_2	**Net_20**	0	**0.3120**	0.000	**1.40**		**-2.01**	100
Net_2	**Net_30**	0	**0.3112**	0.000	**1.70**		**-2.22**	100
Net_2	**Net_10**	0	**0.3019**	0.000	**1.17**		**-1.85**	100
Net_2	**Net_29**	0	**0.2934**	0.000	**1.37**		**-1.91**	100
Net_2	**Net_23**	0	**0.2850**	0.000	**1.32**		**-1.72**	100
Net_2	**Net_21**	0	**0.2638**	0.000	**1.92**		**-2.15**	100
Net_2	**Net_19**	0	**0.2553**	0.000	**1.76**		**-1.85**	100
Net_3	**Net_3**	296.611	**8.0109**	1.325		12.47		
Net_3	**Net_40**	0	**0.8060**	0.000	**2.77**		**-4.94**	100
Net_3	**Net_2**	0	**0.3957**	0.000	**1.69**		**-2.78**	100
Net_3	**Net_22**	0	**0.3864**	0.000	**1.26**		**-2.41**	100

Change Rise Time

1. Use the SHIFT key and mouse to select multiple rows.
2. Use the CTRL key and mouse click to select the top cell, Tr (ps).
3. In the highlighted cell change the Rise Time to the desired value.
4. Click **Enter**. The values are changed.

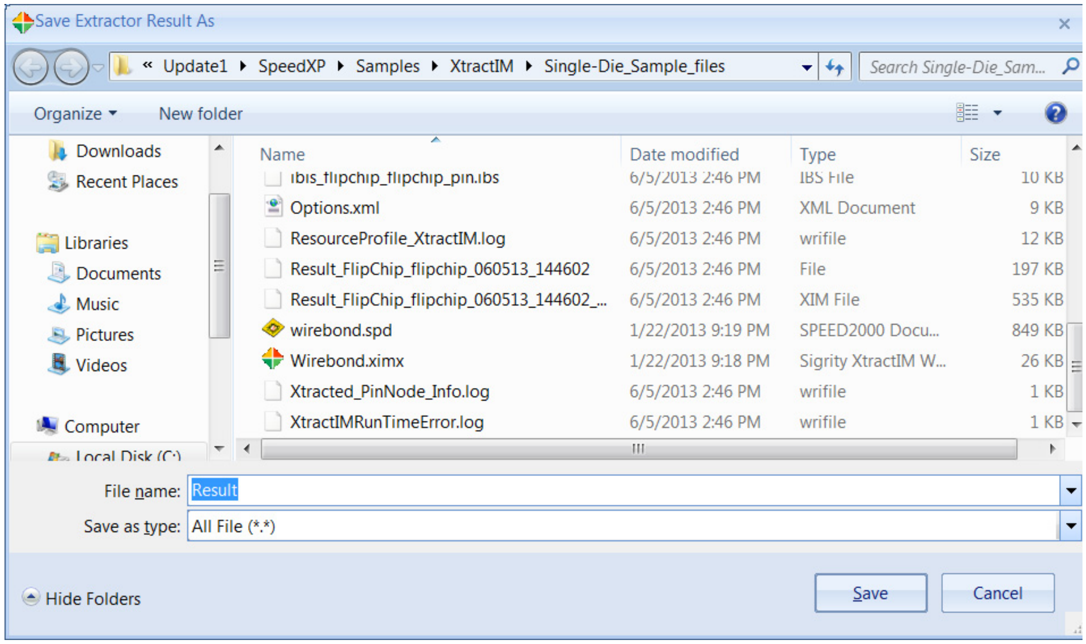
Upon exiting the window, a pop-up window appears. You are asked if you want to save your changes.



Save Simulation Results

1. Click **Save Results** in the **Workflow** pane.

   The **Save Extractor Result As** window appears.



2. Input a file name.

3. Click **Save**.

   The results are saved in a binary file. The file is named:

   result_spd_file_name.xim

4. (Optional) Click **Cancel** if you do not want to save the results in the file name you input in earlier steps.

Simulation Output Files

The **result** and **result*.xim** files save all the output data including Summary, SPICE circuit and the two package model files. The SPICE file and two IBIS files are automatically saved by the tool.

The result file is created automatically when the simulation is finished. The output files can be viewed on the on hard disk.

- **One Crosstalk file in Excel Format** — Signal_Xtalk.csv for signal net crosstalk
- **One IBIS Package Model File** — *.pkg file. Both L and C include coupling elements
- **One Pin Model in Excel Format** — *.csv file includes each signal net length, self-R, self-L, self-C, and time delay. No coupling elements are included
- **One Pin Model in IBIS Format** — *.ibs file including each signal net self-R, self-L, and self-C. No coupling element is included
- **One Summary Content in Excel Format** — *.csv file includes RLC Full Matrix
- **Three Segment RLC in Excel Format** — Segment RLC of each metal layer with *.csv files
- **Two SPICE Circuit Files** — Pi-model is named *.ckt and the T-model is named *_t.ckt

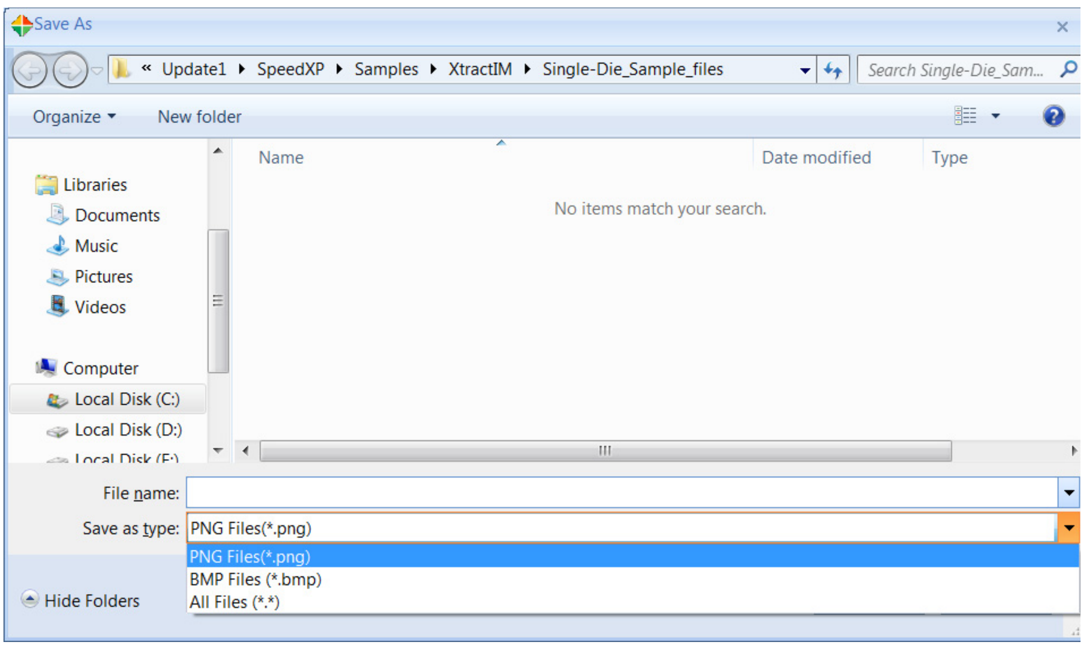
NOTE!	In model (*.ckt, *.pkg, and *PinModel.csv) extraction, XtractIM uses all ground nets as reference for both power and signal nets. The *.ibs model is for signal nets only. In *.ibs model extraction, by IBIS definition, XtractIM uses all ground and power nets as ideal reference to get R, L, and C for each signal net.

Output Result Displays with Images (PNG or BMP Format)

1. Select

   Workspace > Export to Image File

   The **Save As** window opens.

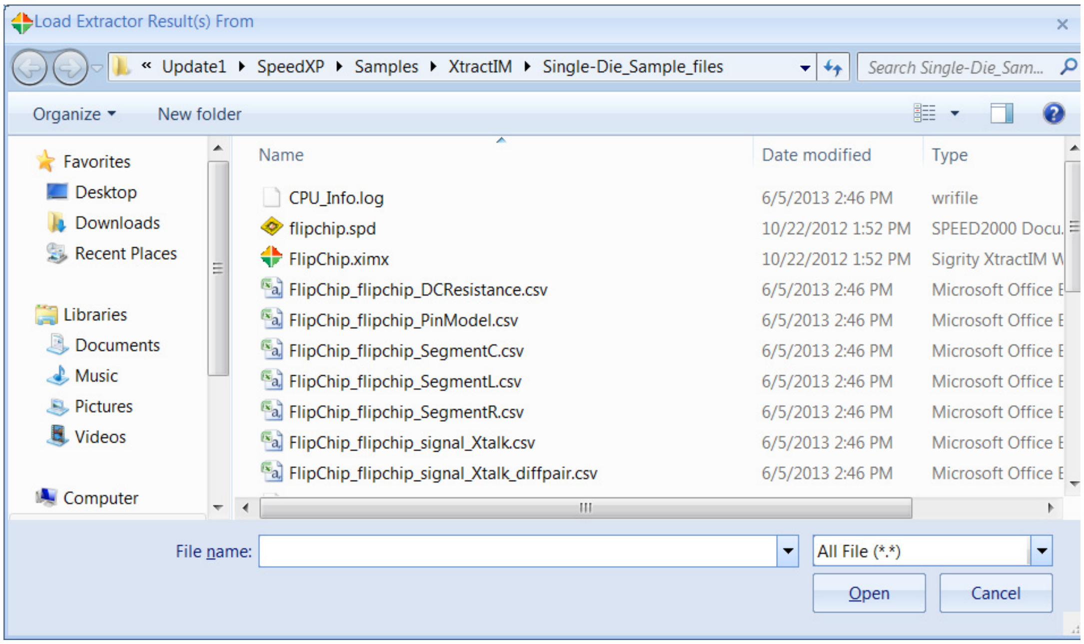


2. In the new window select the file type (.png or .bmp).
3. Click **Save**. The image file is saved.

Load in Saved Results

1. To load the saved results, click **Load Results** in the **Workflow** pane.

   The **Load Extractor Result(s) From** window opens.



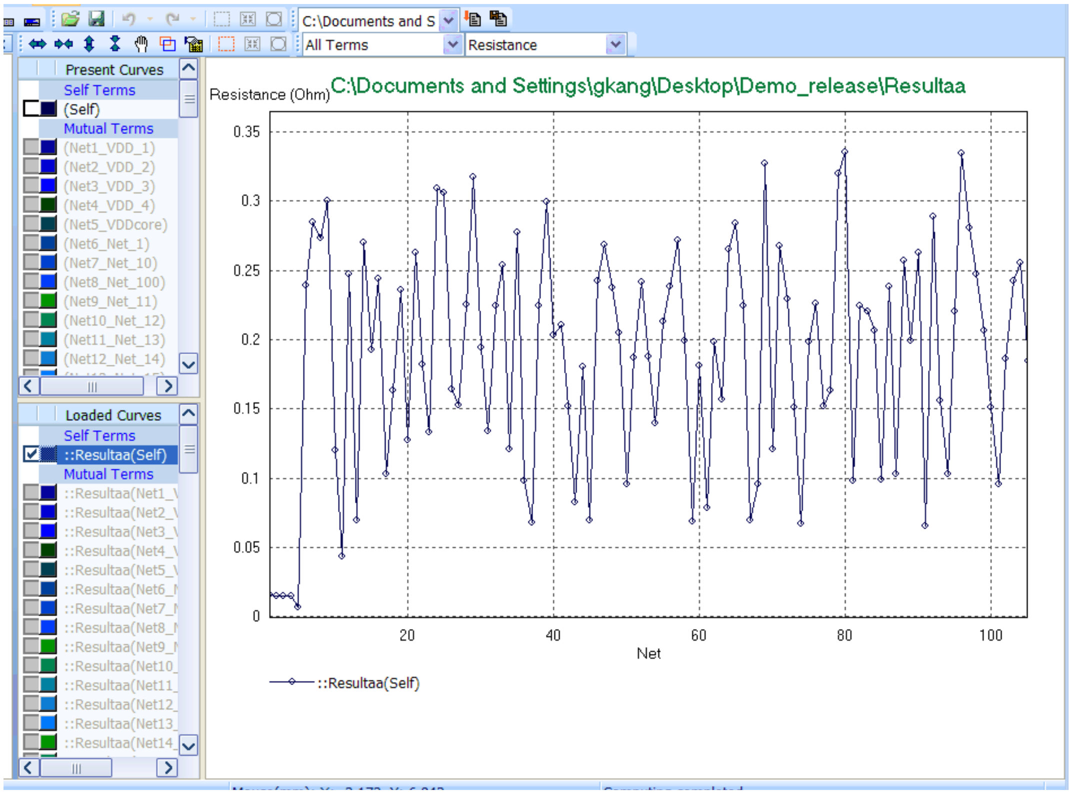
2. Browse to select the desired result file.
3. Click **Open.**

   The result file is loaded.

4. View present results or the loaded results.
   - **Loaded Curves** — Shows the loaded results
   - **Present Curves** — Present extracted results
5. To unload a loaded result:

   Unload Extractor Result

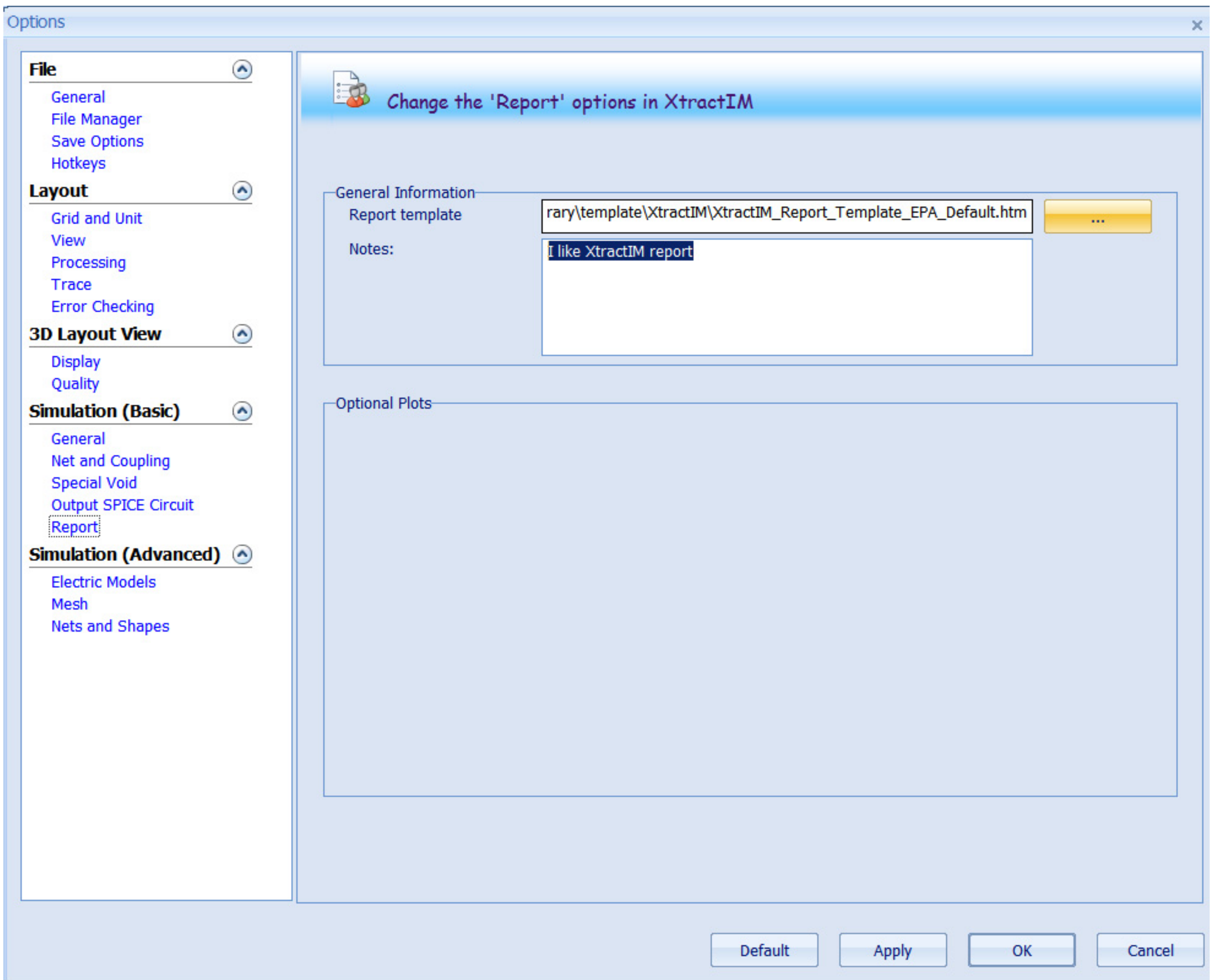
Resistance Results



Simulation Report with .htm / .html Format

1. Click **Report** in the **Workflow** pane.

   The **Options** window appears.



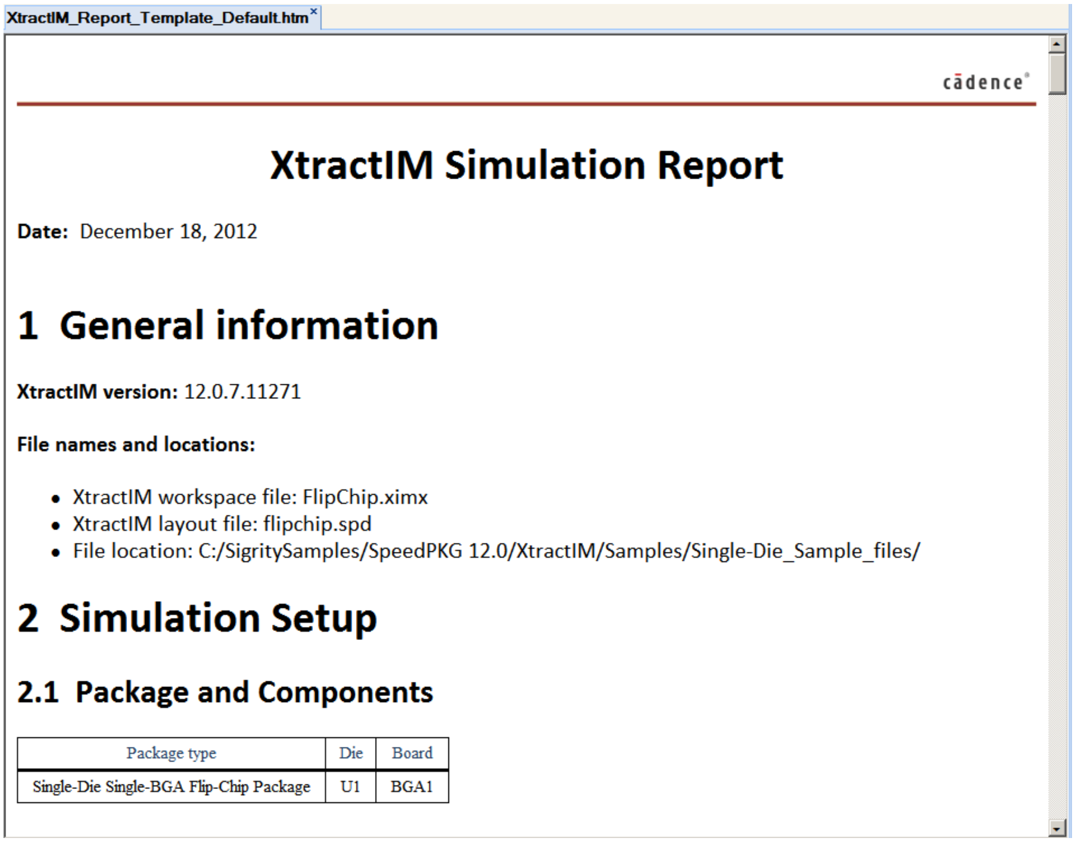
2. Browse to select the Report Template from

   <INSTALL_DIR>\SpeedXP\Library\template\XtractIM\

3. Input notes, if desired.

4. Click **OK**.

The report file is created.


XtractIM_Report_Template_Default.htm

cādence

XtractIM Simulation Report

Date: December 18, 2012

1 General information

XtractIM version: 12.0.7.11271

File names and locations:

- XtractIM workspace file: FlipChip.ximx
- XtractIM layout file: flipchip.spd
- File location: C:/SigritySamples/SpeedPKG 12.0/XtractIM/Samples/Single-Die_Sample_files/

2 Simulation Setup

2.1 Package and Components

Package type	Die	Board
Single-Die Single-BGA Flip-Chip Package	U1	BGA1

Lesson Five: Batch Mode Simulation

1. To run a simulation in Batch Mode, select:

   Start > Run

2. Change to the directory where the XtractIM.exe file is located.
3. Upon completing the simulation, all output files are automatically saved in the same directory as the *.spd file.
   - *.ckt files
   - *.ibs files
   - *.pkg files
   - .csv files

Batch Mode Example

If you want to use the project (.spd file) defaulted in the workspace file (.xml file), enter

XtractIM -b “Full_path_toMy_xml_File *xml filename*”

If you want to use a different project file other than the one in the .xml file, enter

XtractIM -b “Full_path_toMy_xml_File\xml filename” “Full_path_toMy_xml_File *new_spd-file-name*”

Lesson Six: Partial Inductance Report for Power/Ground Nets

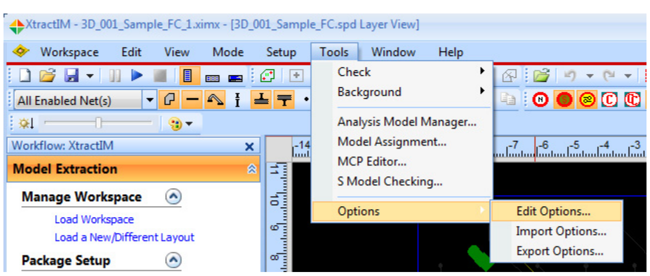
Introduction

Net-based partial inductance of power and ground nets is an optional output result for single-die single-BGA packages. It is used to evaluate the relative power and ground network design quality, while loop inductance of power and ground net pair provides meaningful electrical performance.

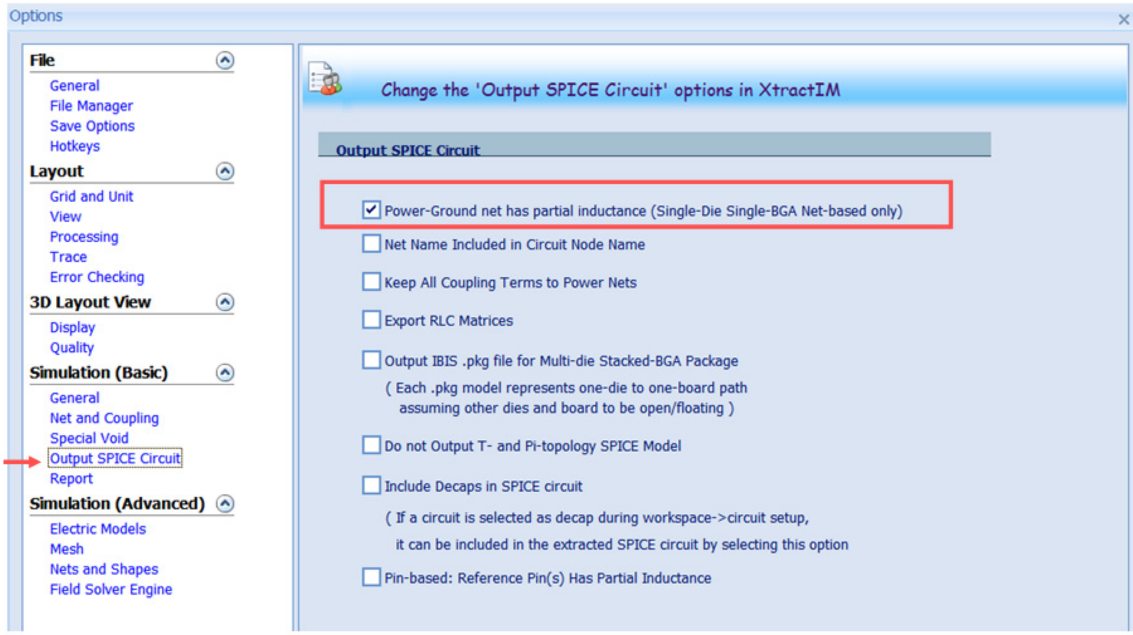
In the **SPICE/IBIS Model** section of **View/Export Results**, three partial inductance reports are available.

- Partial RLC for Power and ground nets reported with .csv format
- T-topology SPICE model with Power-ground nets having partial inductance
- Pi-topology SPICE model with Power-ground nets having partial inductance

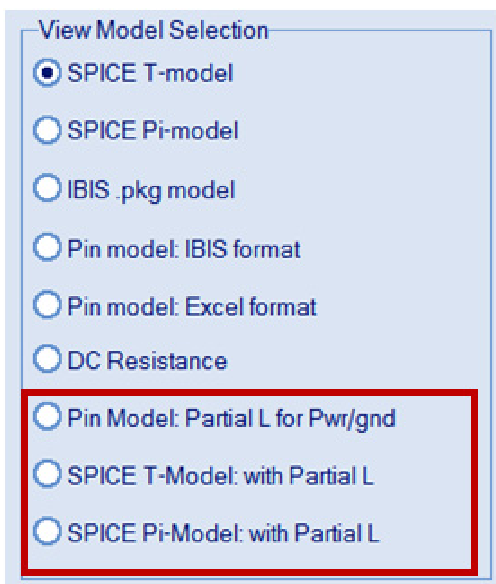
1. Choose Tools > Options > Edit Options….



2. Click **Output SPICE Circuit** in the **Options** window.
3. Check **Power-Ground net has partial inductance (single-Die Single-BGA Net-based only)**.



In the **Model Selection** section, partial inductance reports for power and ground nets are also available.



LESSON SEVEN: 3D-EM OPTIONAL FIELD SOLVER IN XTRACTIM

Introduction

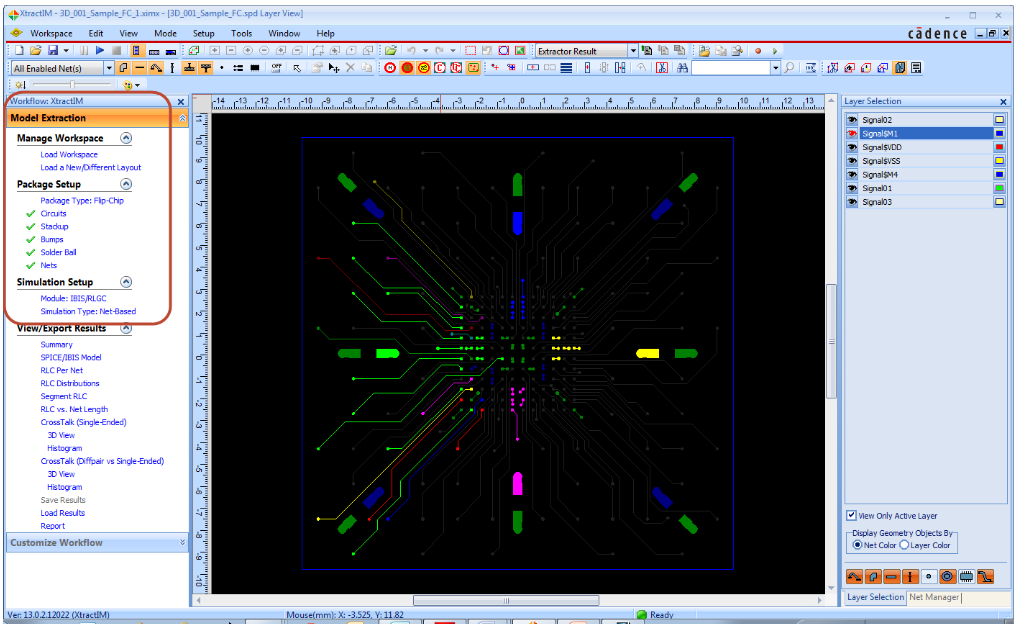
Cadence 3D-EM is an optional field solver for XtractIM. It's targeted to solve simpler packages such as lead frame package and two-layer BGA package with higher accuracy. The workflow is totally consistent with the existing hybrid solver bases.

The integration of 3D-EM and XtractIM is only applied for single-die single-BGA and single-die leadframe for Net-based RLGC/IBIS model at single-frequency.

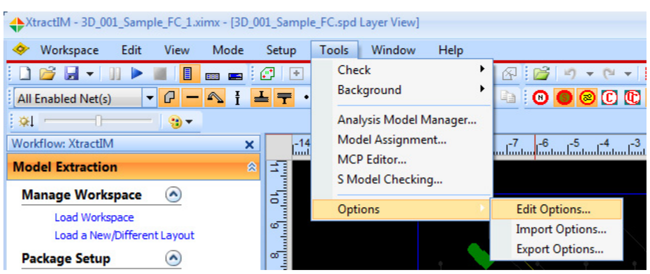
For better performance, it is recommended that you select no more than 20 nets with at most four layers.

Enabling 3D-EM Field Solver

1. Make sure all the workflow setup is completed with green check-mark.

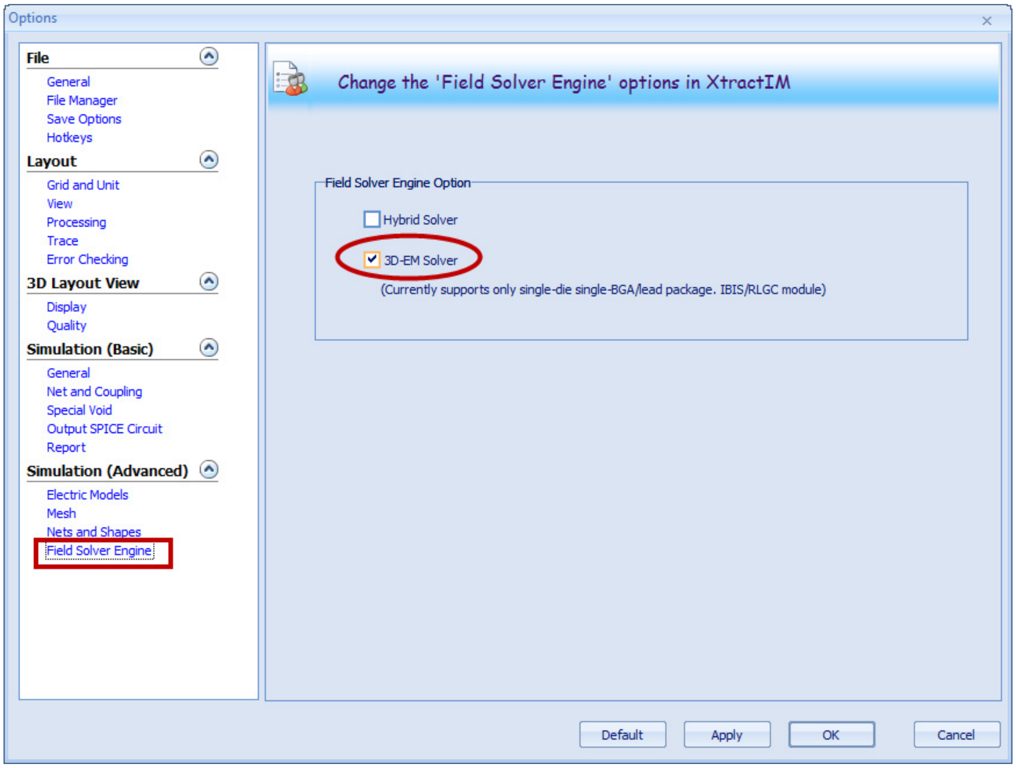


2. Choose Tools > Options > Edit Options….

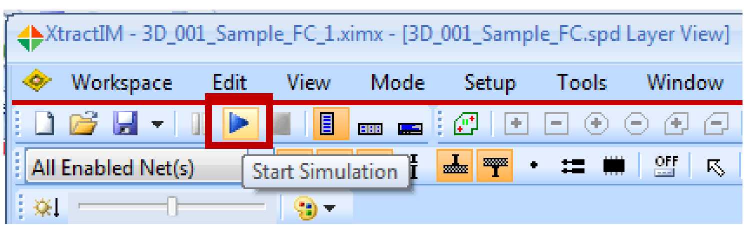


3. Click **Field Solver Engine** under **Simulation (Advacned)** in the **Options** window.

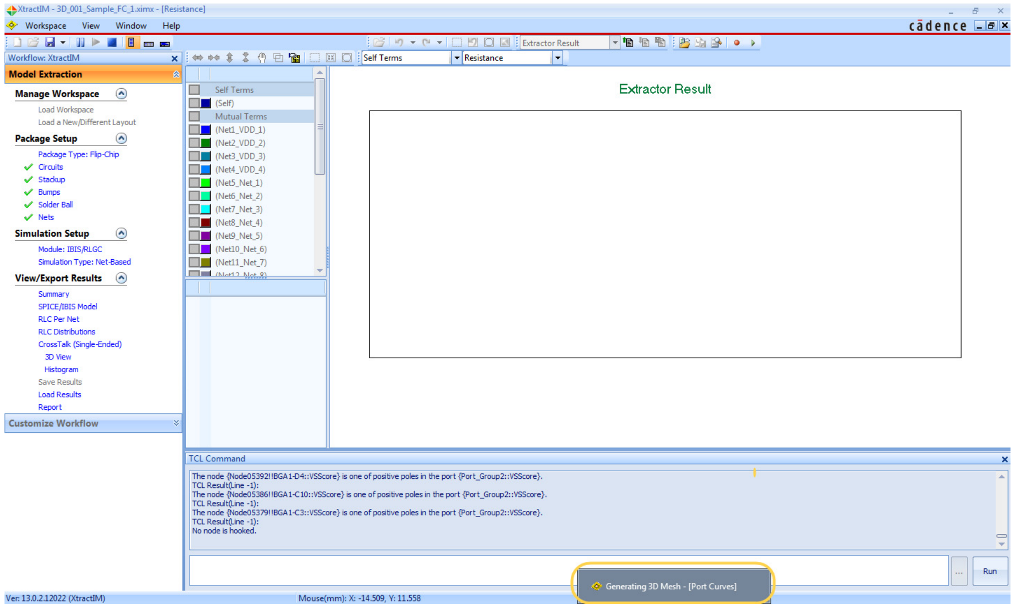
4. Check **3D-EM Solver** in **Field Solver Engine Option**.



5. Click **OK**.
6. Click the **Start Simulation** button to run simulation.



3D-EM is automatically launched as solver, and RLC result is obtained.



A temp3DFEM.spd file is generated for 3D-EM simulation. This SPD file has all settings required for 3D-EM, including:

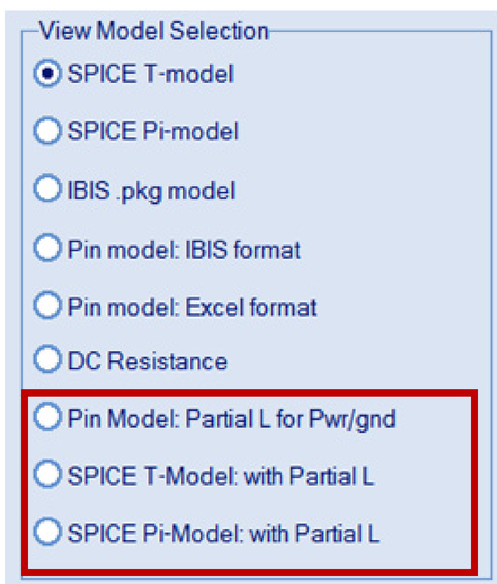
- Port defined
- Frequency settings
- Solver options
- Geometry settings

You can use this SPD file for advanced 3D-EM simulation.

Results Display

- With 3D-EM field solver enabled, the following result is available.

- In the SPICE/IBIS Model, partial inductance reports for power and ground nets are also available.



Chapter 4

RLGC Module: Net-based Simulation of Multi-die Stacked-BGA Packages

This chapter takes you through the steps to use the XtractIM tool in the simulation of the multi-die stacked BGA Package.

PREPARE FOR SIMULATION

Collect this information before you begin the simulation.

- Make sure your files have been translated into SPD format.
- Have the Stackup information ready.
- Have the Bump and Solderball diameters, length, heights and conductivity ready.

LESSON ONE: SIMULATION SETUP

Simulation Overview

1. Load an existing file (.xml file) or open a layout file (.spd file).
2. Select a package type: Wirebond or Flip-Chip, single BGA or stacked BGA.
3. Setup the circuits: **Die-circuit** or **Board circuit**.
4. Setup the **Stackup**. Set parameters for the Bump / Solderball medium layer.
5. Set the **Bump** data if you choose a Flip-Chip package.
6. Set the **Solder Ball** data. Select the nets for extraction.
7. Setup extraction frequency and capacitance / inductance output control.
8. Save the workspace.
9. Save Layout files.
10. Run the simulation.

Setup Package Simulation

Simulation setup is similar to the setup procedures described in ***IBIS/RLGC Module: Net-based Simulation of Single-die Single-BGA Packages***. The only difference is in three steps.

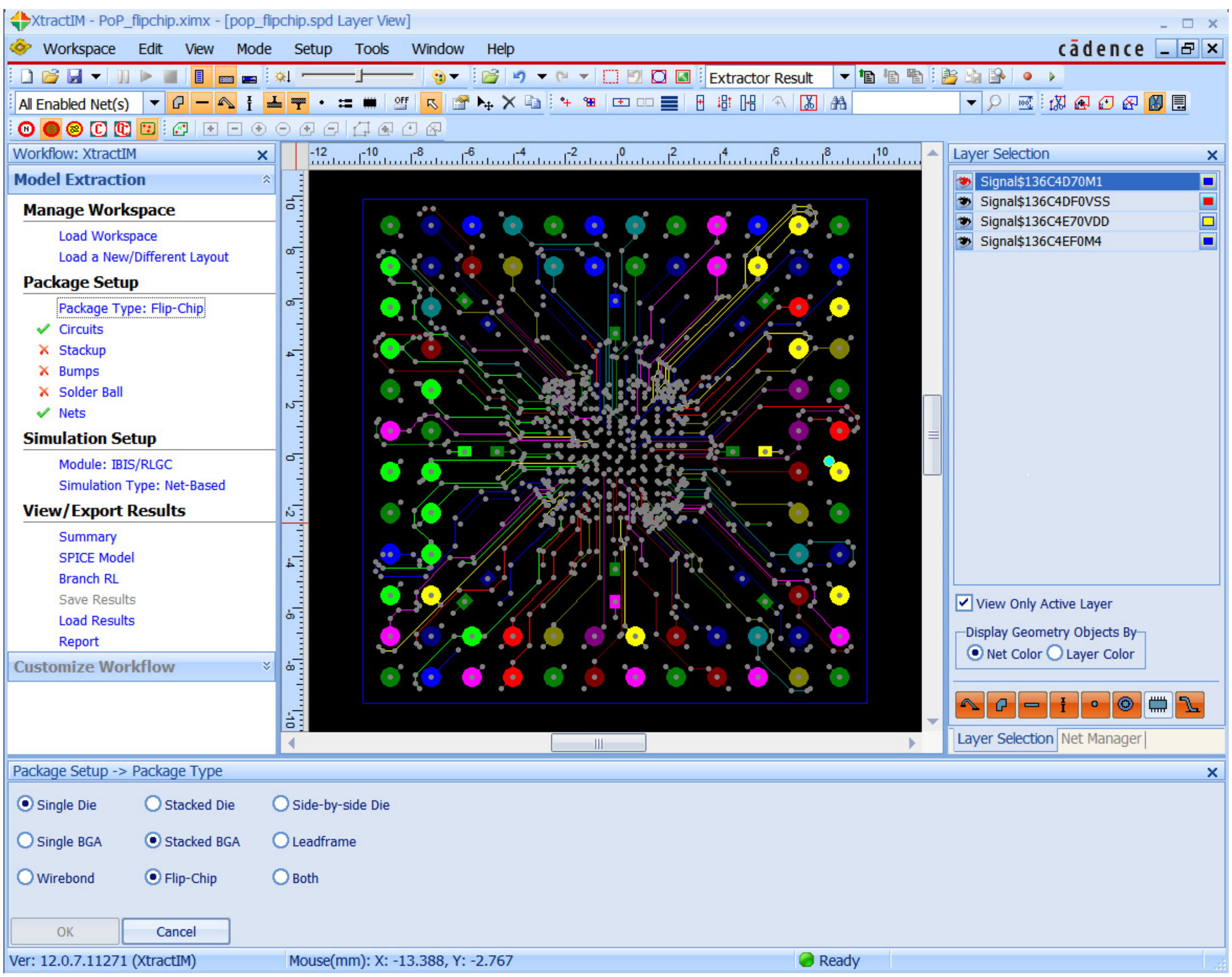
- Select a Package Type
- Setup Circuits
- Setup Stackup

Setup Package Type

1. Click on the appropriate radio button from these options:
   - Flip-Chip
   - Side-by-side Die
   - Single BGA
   - Single Die
   - Stacked BGA
   - Stacked Die
   - Wirebond

   or select both Wirebond and Flip-Chip for various packages.

   The default package type is **Single Die, Single BGA, Wirebond**.
2. Click **OK** to save your selection.
3. Click **Cancel** if you wish to start over and re-enter your settings.

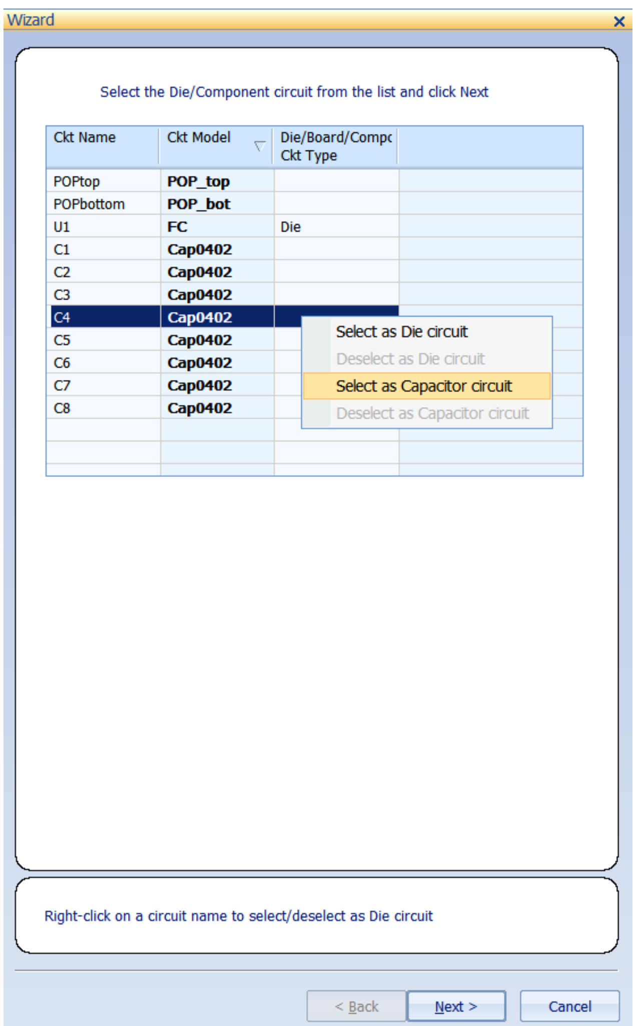


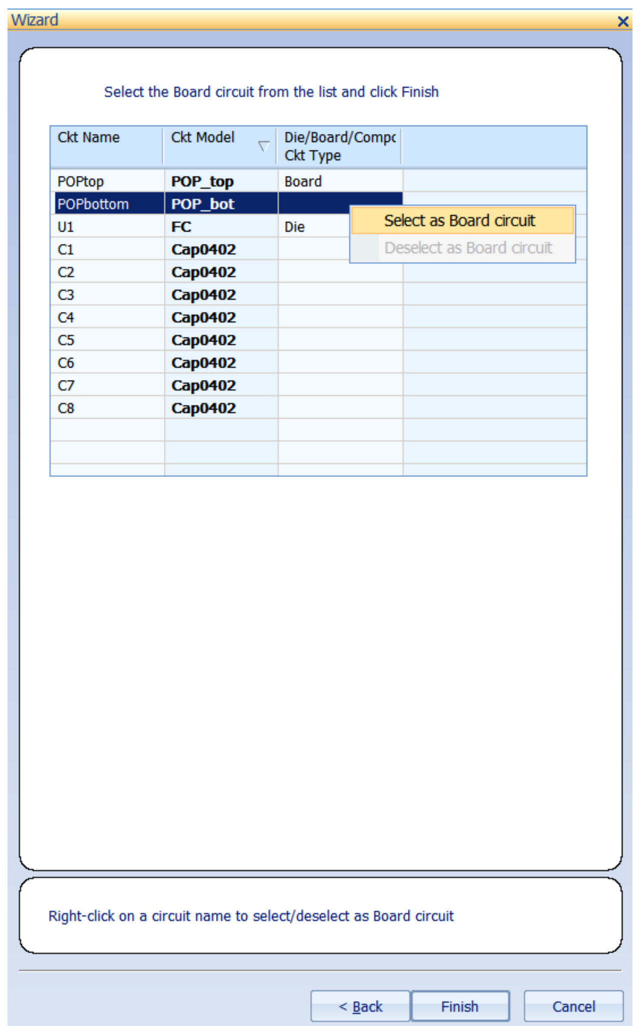
Setup Circuits

1. Click on **Circuits** in the Workflow pane to setup the Circuits data for a Flip-Chip package. A new pane opens.
2. Right-click on the desired circuit.
3. Select it as **Die** circuit (you can also select it as **Capacitor** circuit).
4. Click **Next**.
5. Right-click on another desired circuit.
6. Select it as a **Board** circuit.
7. To set up the second as a Board circuit, choose:

   Select as Board circuit

8. Click **Finish** to finish the setup.
9. Select the multi-die circuit if the package has multi-dies.





Setup Stackup

1. Click on **Stackup**. The Stackup window opens.
2. In the **Layer Name** column click on:

   Signal$Bottom layer (Signal$136C4EF0M4)

   A pop-up window opens.
3. Select

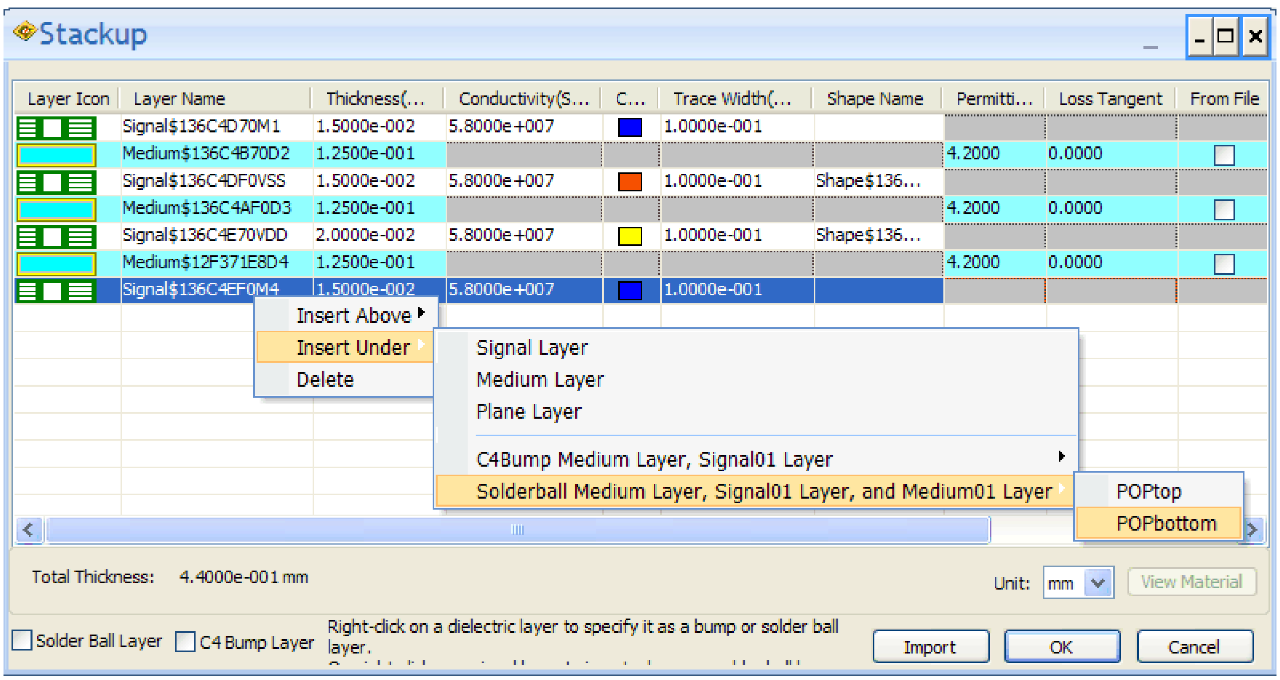
   Insert Under

   A second pop-up window opens and displays layer options.
4. Click:

   Solderball Medium Layer, Signal01 Layer, and Medium01 Layer

   A menu opens.
5. To insert all layers under the Signal$Bottom layer (Signal$136C4EF0M4)

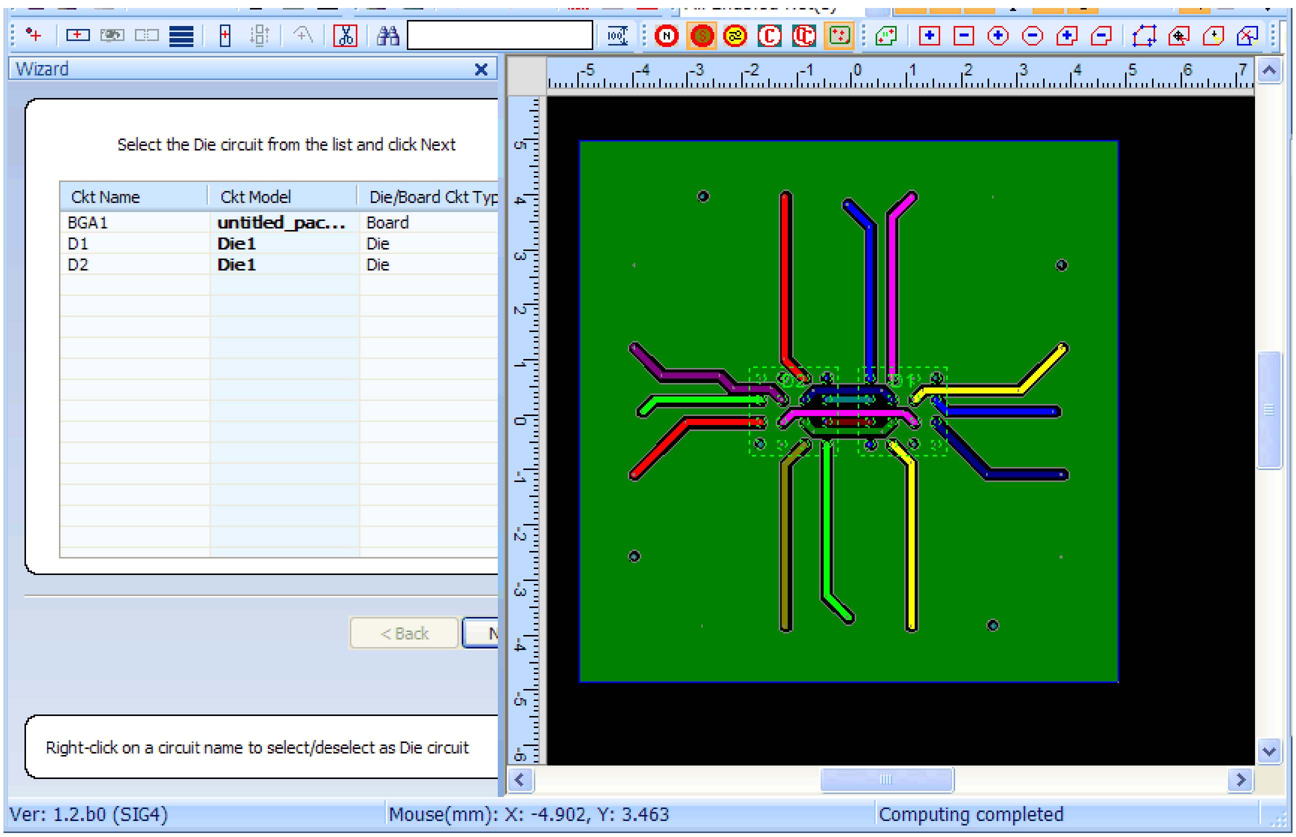
   POPbottom



Setup Bump and Solder Ball Medium Layers

The newly-added signal layer is found at the end of the solderball.

1. Right-click on the **Signal$Top** layer.
2. Insert a **Bump Medium Layer** and a **signal layer** above Signal$Top. The added signal layer is the end of the Bump.
3. Return to the **Stackup**.
4. View the Bump and Solder Ball Medium Layers as they are created.
5. Edit the medium parameters for Bumps, if desired.
6. Edit the solderball, if desired.
7. Edit the PCB medium layers, if desired.



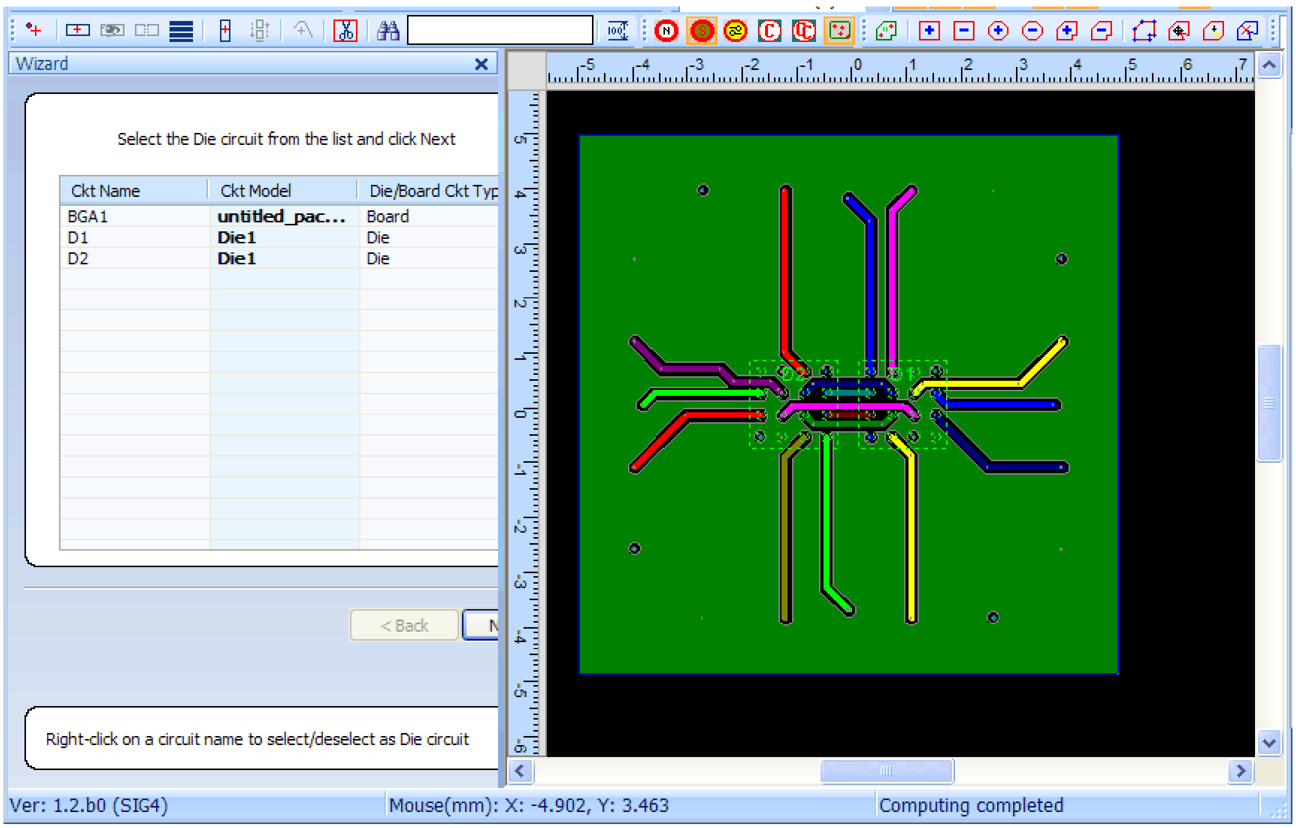
Setup Bumps to a Side-by-Side Die Package

If the side-by-side Die Flip-Chip package has the same Bump height in all die circuits, it is treated as a **distributed die package**.

If the heights are different, it is treated as a stacked die package.

1. Add one Bump layer.
2. Open the **Stackup** for the distributed Bump die package.

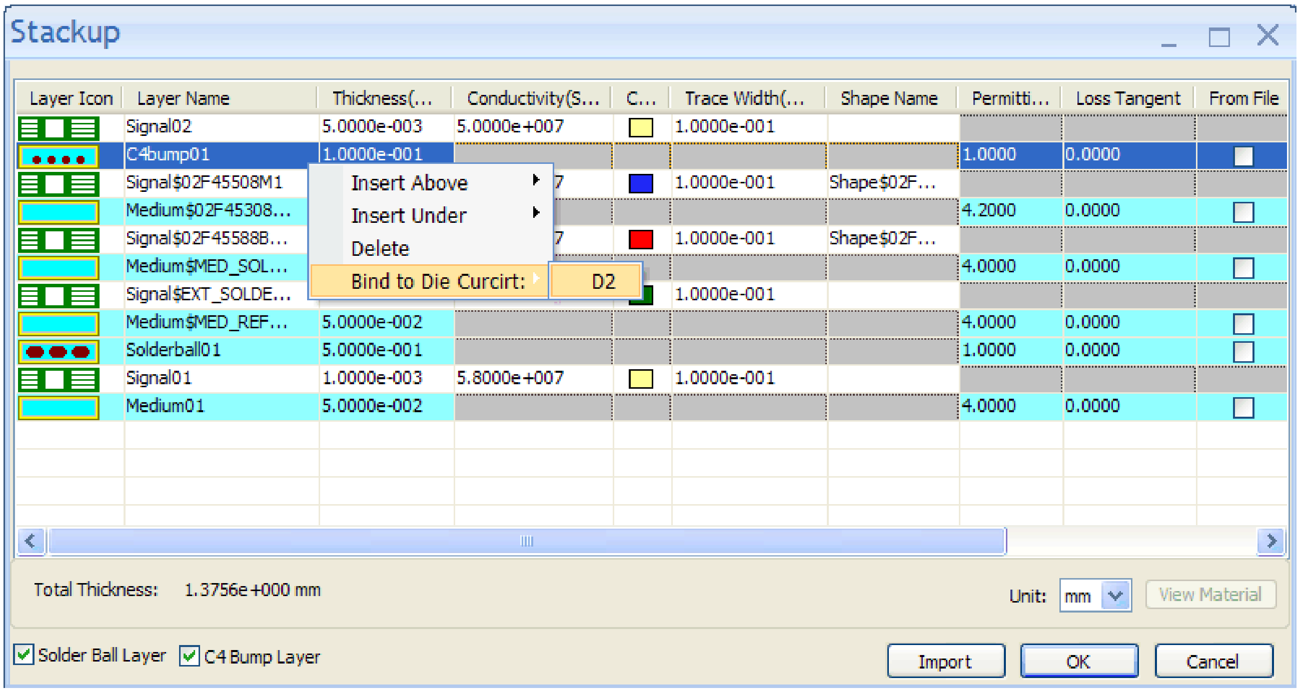
   You can bind them together to add the same Bump medium layer for each die circuit.



3. After adding the Bump for D1, select:

   Bind to Die Circuit: D2

   Both D2 and D1 have the same layer thickness.

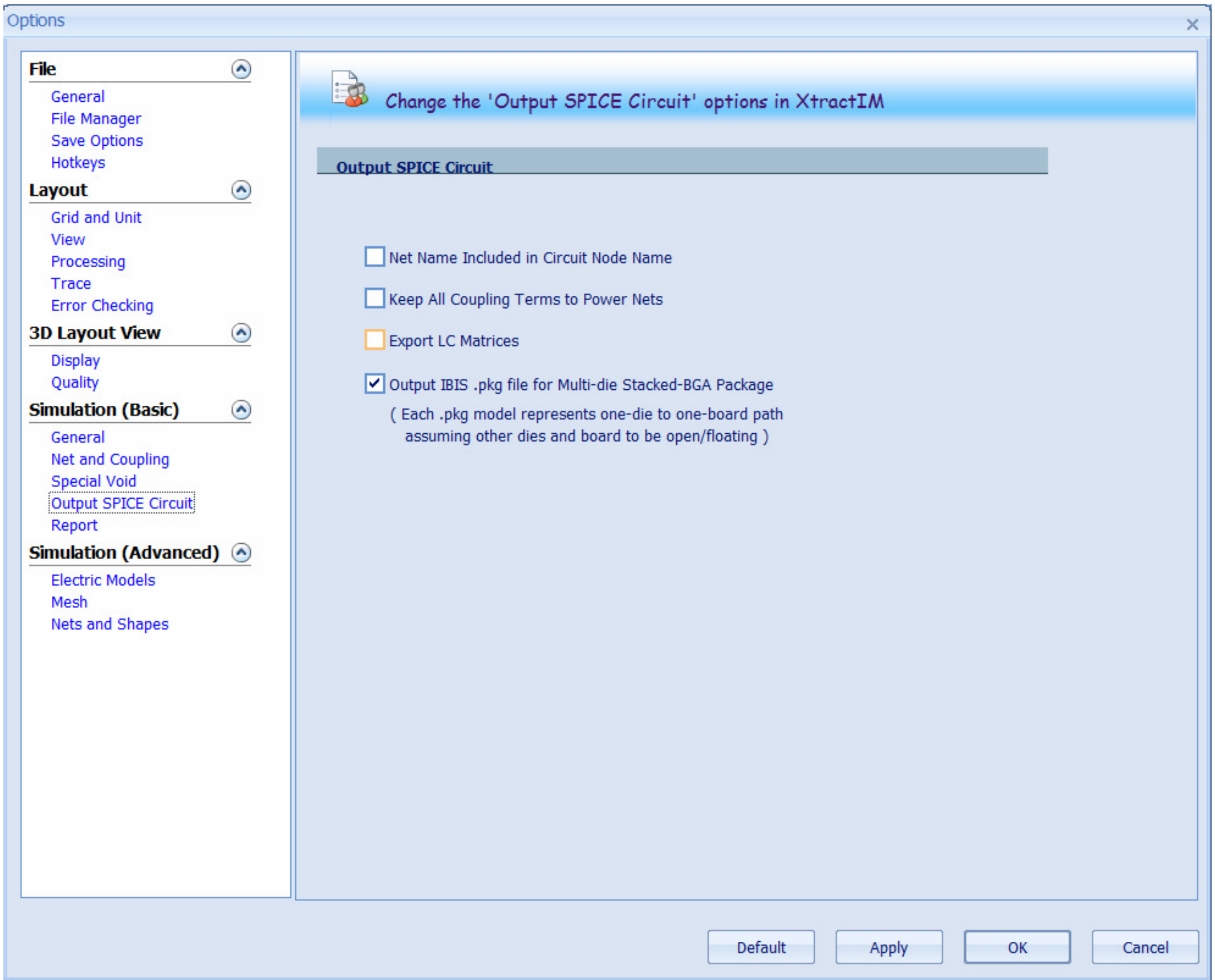


Output IBIS .pkg Model for Each Die-to-Board Path

1. To output IBIS .pkg model for each die-to-board path, select

   Tools > Options > Edit options...

   The **Options** window open.



2. Click **Output SPICE Circuit** under **Simulation (Basic)**.
3. Select the **Output IBIS .pkg file for Multi-die Stacked-BGA Package** checkbox.
4. Click **OK**.

Lesson Two: Save Work Areas

Layout File

1. Click on the workspace.
2. Click on the **Open Layout File** icon (**green**) to open the project file.
3. Click on the **Save Layout File** icon (**green**) to save the project file.
4. Use **Save as** to save the layout file under a different name.

NOTE!	Saving the workspace does not automatically save the .spd file. Saving the.spd file does not automatically save the workspace.

Workspace

1. Click on the workspace. The workspace opens. Note the workspace toolbar.
2. Click on the **Save Workspace** icon (**yellow**) in the toolbar.
3. To save the workspace under a different name, select:

   Save as

Lesson Three: Run the Simulation

Click on the **Play** button  at the top of the window to start the extraction (simulation).

XtractIM only extracts RLCG for the net which has at least one pin at the Die side and at least one pin at the board side.

At the beginning of the simulation, if some nets have Die-Board mis-match, a pop-up window opens. You are asked to select the next action.

- **Continue** — Continue the simulation
- **More Information** — Examine what nets are mis-matched
- **Stop** — Cancel the simulation

Investigate Mis-matched Nets

1. The **More Information** window lists all the mis-matched nets.
2. Investigate the mis-matched nets to see whether it is a special design or a defective design.
3. Decide whether or not to proceed with the simulation.
4. Choose **Continue**, **Stop** or **More Information**.

   If 30 seconds pass and the user has not made a choice; by default, the simulation continues.

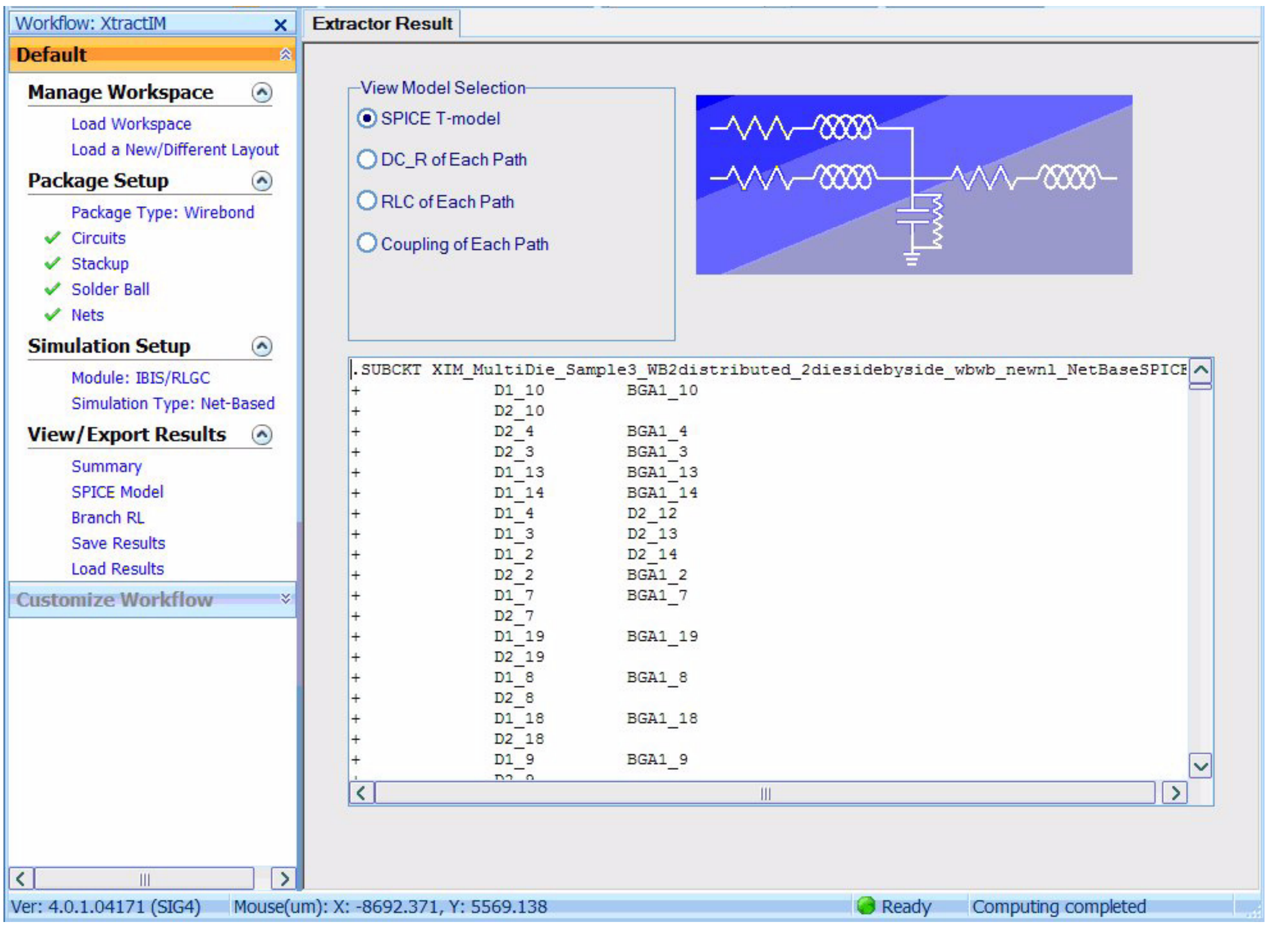
Lesson Four : Observe and Save Simulation Results

XtractIM performs calculations for each net. The calculations include:

- Conductance
- Mutual capacitance with other nets
- Mutual loop inductance
- Resistance
- Self capacitance
- Self loop inductance

Display SPICE Model Results

The illustration in the right corner shows the detail of the Circuit topology for a 2-die BGA package.



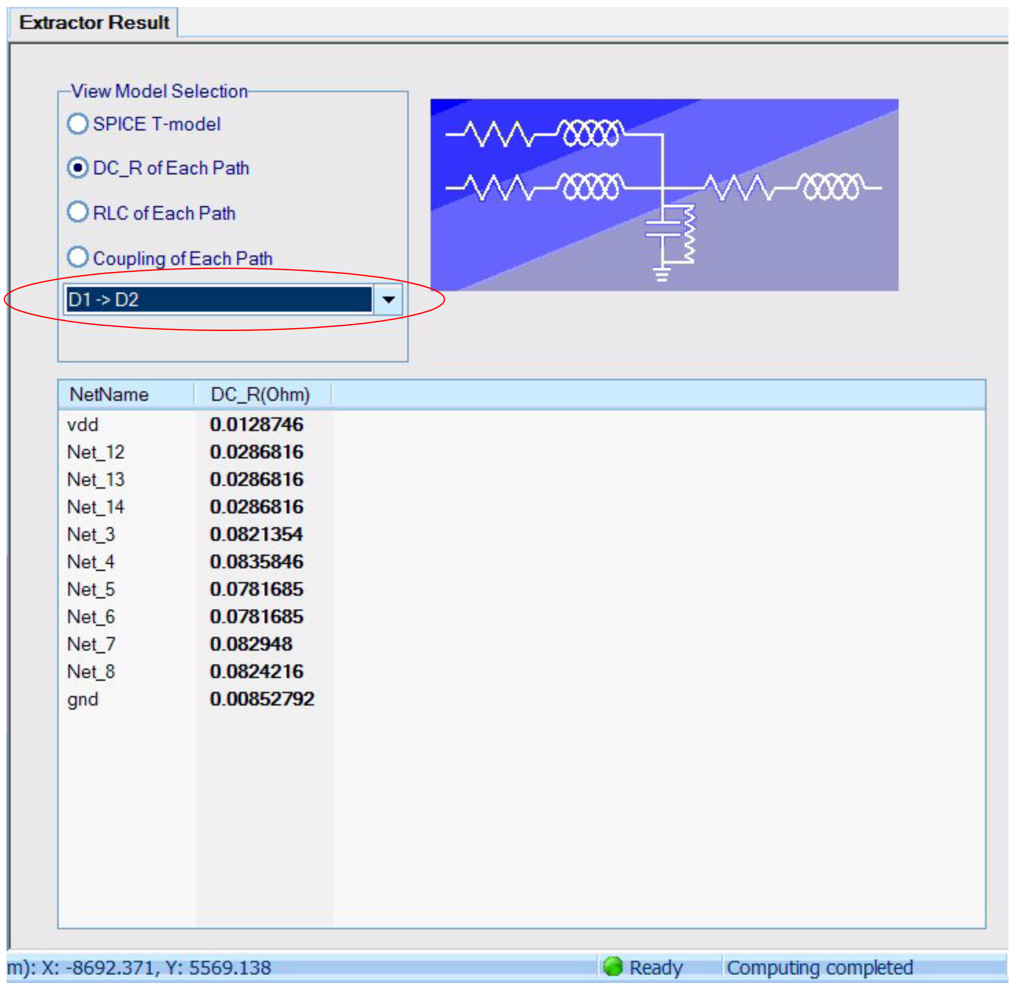
View DC Resistance

1. To view the DC Resistance of Power, Ground and Signal Nets of each circuit-to-circuit path, click:

   DC_R of Each path

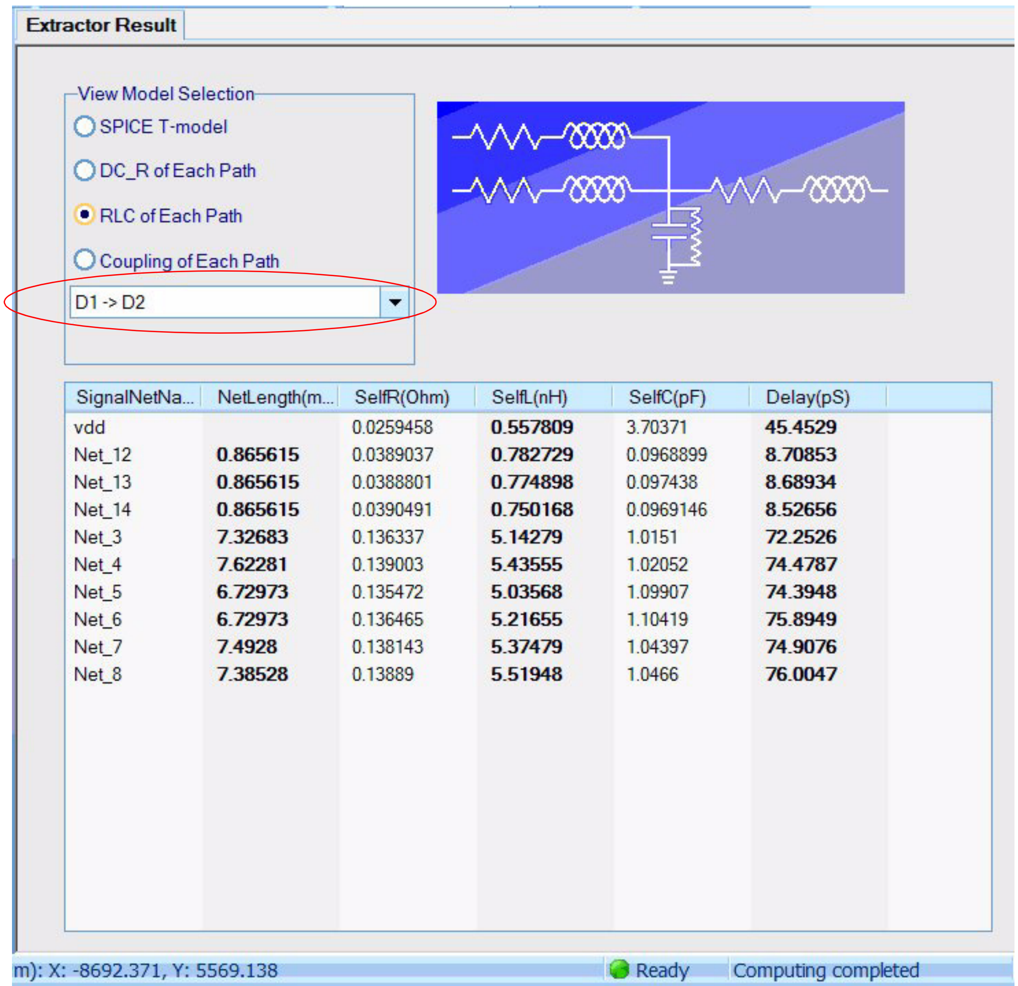
2. In the drop-down pane make a selection for a circuit-to-circuit path. In the following example the selection is:

   D1 > D2



Display RLC of Each Path

1. To view the **Net Length**, **Self R**, **Self L**, **Self C**, or **Delay** of a Signal Net., click on RLC of Each Path
2. Use the drop-down pane to select a circuit-to-circuit path.

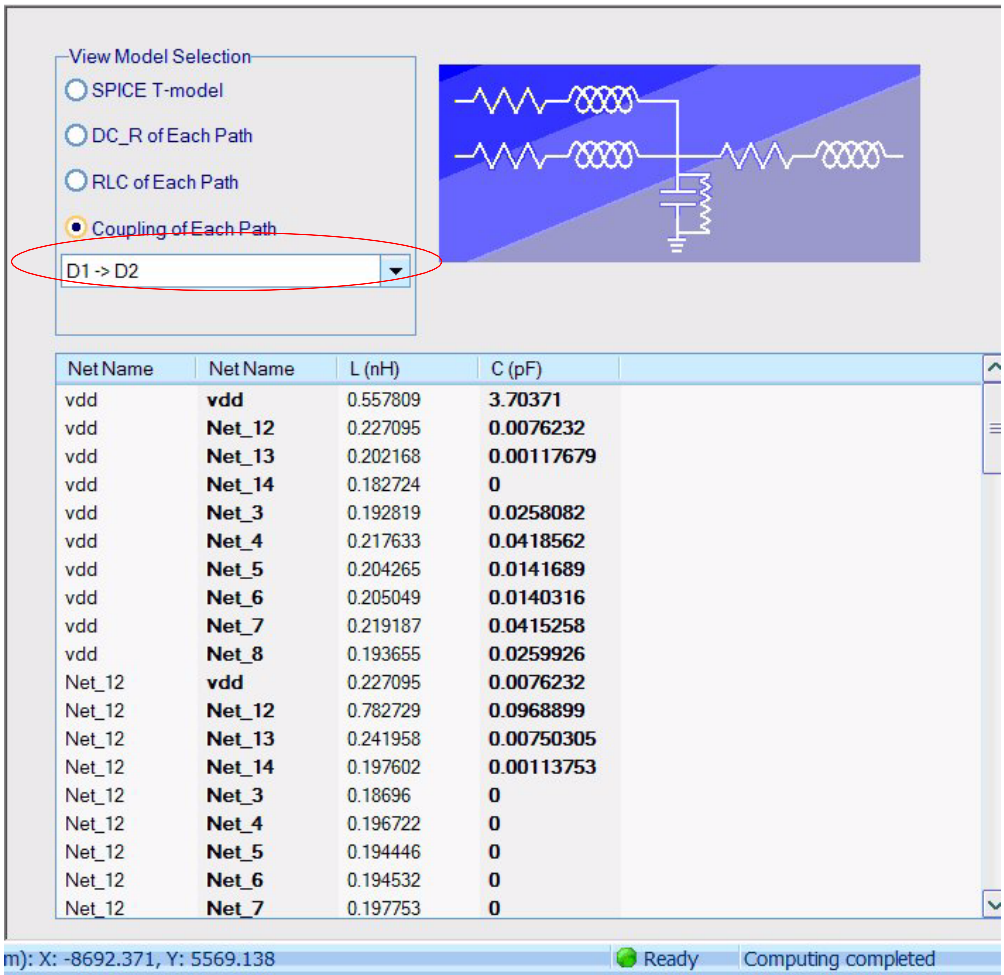


Display Coupling of Each Path

1. To view **Mutual L**and **Mutual C** among nets along the path, click:

   Coupling of Each Path

2. Use the drop-down pane to select the circuit-to-circuit path.



Net Name	Net Name	L (nH)	C (pF)
vdd	**vdd**	0.557809	**3.70371**
vdd	**Net_12**	0.227095	**0.0076232**
vdd	**Net_13**	0.202168	**0.00117679**
vdd	**Net_14**	0.182724	**0**
vdd	**Net_3**	0.192819	**0.0258082**
vdd	**Net_4**	0.217633	**0.0418562**
vdd	**Net_5**	0.204265	**0.0141689**
vdd	**Net_6**	0.205049	**0.0140316**
vdd	**Net_7**	0.219187	**0.0415258**
vdd	**Net_8**	0.193655	**0.0259926**
Net_12	**vdd**	0.227095	**0.0076232**
Net_12	**Net_12**	0.782729	**0.0968899**
Net_12	**Net_13**	0.241958	**0.00750305**
Net_12	**Net_14**	0.197602	**0.00113753**
Net_12	**Net_3**	0.18696	**0**
Net_12	**Net_4**	0.196722	**0**
Net_12	**Net_5**	0.194446	**0**
Net_12	**Net_6**	0.194532	**0**
Net_12	**Net_7**	0.197753	**0**

Summary of the Extracted Results

1. Click on **Summary** to open a brief summary of the extracted R, L, and C in the SPICE circuit.
2. View the maximum and minimum values, and other information in the **Brief Summary** window.

Extractor Result

```
[Version]                       4.0.1.04171
[Date]                          04/16/2010
[Package Name]                  C:\Users\gkang\Desktop\Samples\Multi-die_Sample_
[Description]                   C:\Users\gkang\Desktop\Samples\Multi-die_Sample_
[Nets Extracted]                16
[Frequency of Extraction]       30MHz
[Max R(mOhm)]                   62.2883
[Min R(mOhm)]                   7.26778
[Max self-inductance L(nH)]     2.40819
[Min self-inductance L(nH)]     0.356281
[Max self-capacitance C(pF)]    3.70371
[Min self-capacitance C(pF)]    0.0968899
```

Branch RL and Total C

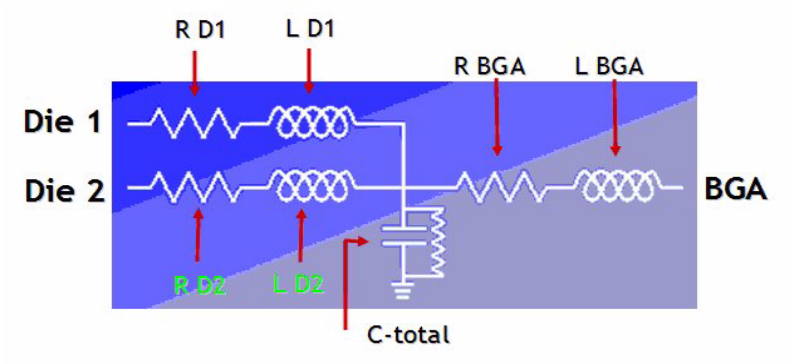
Click **Branch RL** to view the display the **Branch R**, **L**, and total **C** of each net.

If a branch is missing from a net, the cell is left empty.

Extractor Result

Net Name	C-total (pF)	R D1(Ohm)	R D2(Ohm)	R BGA1(Ohm)	L D1(nH)	L D2(nH)	L BGA1(nH)
vdd	**3.70371**	0.0189957	**0.0190755**	0.00726778	**0.502131**	0.494327	**0.360268**
Net_0	**0.586283**		**0.0499985**	0.0144941		1.34022	**0.898667**
Net_1	**0.519641**		**0.0467442**	0.0123515		1.23658	**0.78594**
Net_10	**0.507267**	0.046825		0.012005	**1.33644**		**0.779702**
Net_11	**0.573214**	0.0502395		0.0140962	**1.4453**		**0.889757**
Net_12	**0.0968899**	0.0183871	**0.0502395**		**0.3721**	1.4453	
Net_13	**0.097438**	0.0183802	**0.0183871**		**0.368179**	0.3721	
Net_14	**0.0969146**	0.0184685	**0.0183802**		**0.356281**	0.368179	
Net_2	**0.585876**		**0.0499099**	0.0142809		1.32604	**0.843666**
Net_3	**1.0151**	0.0611592	**0.0589**	0.012166	**2.2961**	1.89191	**0.955124**
Net_4	**1.02052**	0.0622883	**0.0594929**	0.0125821	**2.40819**	1.96941	**1.00588**
Net_5	**1.09907**	0.0607214	**0.0603477**	0.0129294	**2.1364**	2.03507	**1.023**
Net_6	**1.10419**	0.0607857	**0.0610096**	0.0129482	**2.12252**	2.17874	**1.0308**
Net_7	**1.04397**	0.0602726	**0.0612385**	0.012329	**2.08603**	2.26651	**0.99801**
Net_8	**1.0466**	0.0602456	**0.0619516**	0.0123209	**2.07494**	2.39202	**0.998978**
Net_9	**0.5723**	0.050101		0.0138617	**1.42172**		**0.833771**

The figure below illustrates the Branch RL and Total-C for one net of a 2-die 1-BGA package. It is straightforward for multi-die stacked-BGA packages.



Save Results and Output Files

1. Click:

   Save Results

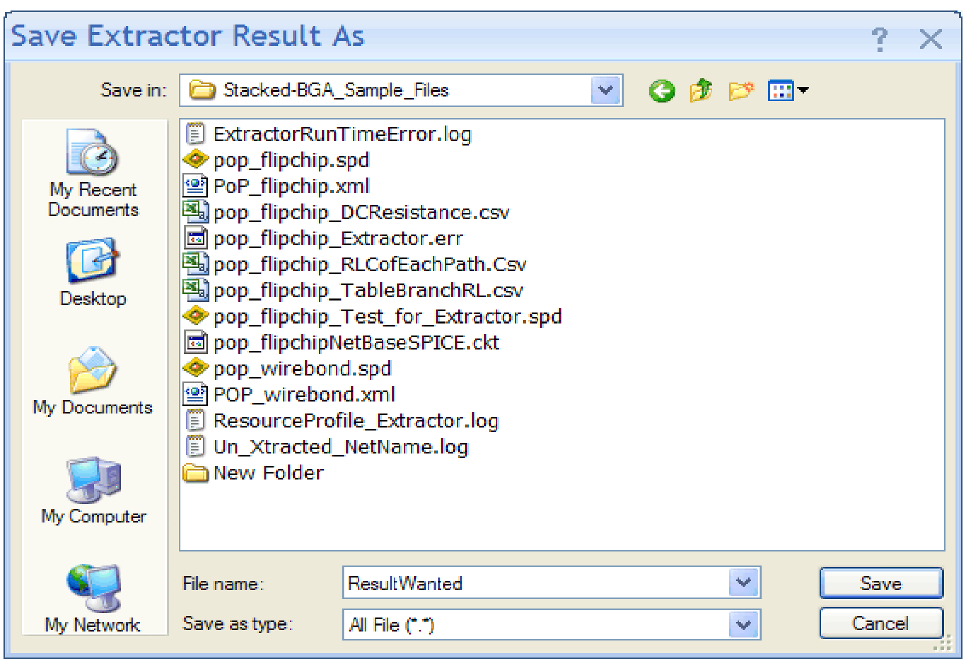
   The **Save Extractor Result** window appears.
2. Enter a name in the **File name** window.
3. Click on **Save**. The results are saved in a binary file named as result and as:

   result_spd_file_name.xim
4. Click on **Cancel** if you do not want to save the results in the file name you entered.
5. Save all the output data including Summary and one SPICE model in the **result** and **result*.xim** files.

The result file is created only when the user chooses to save the output data.

You'll see these output files on hard disk.

- **Branch RL File** — An .csv file including the branch R, L, and total C of each net
- **Coupling of Each Patch** — A .csv file including Mutual L and Mutual C among nets along every circuit-to-circuit path
- **DC R of Each Path File** — A .csv file including each net's DC Resistance
- **One SPICE Circuit File** — Named *.ckt
- **RLC of Each Path File** — A .csv file including each Signal net name, Length, Self R, Self L, Self C and Delay



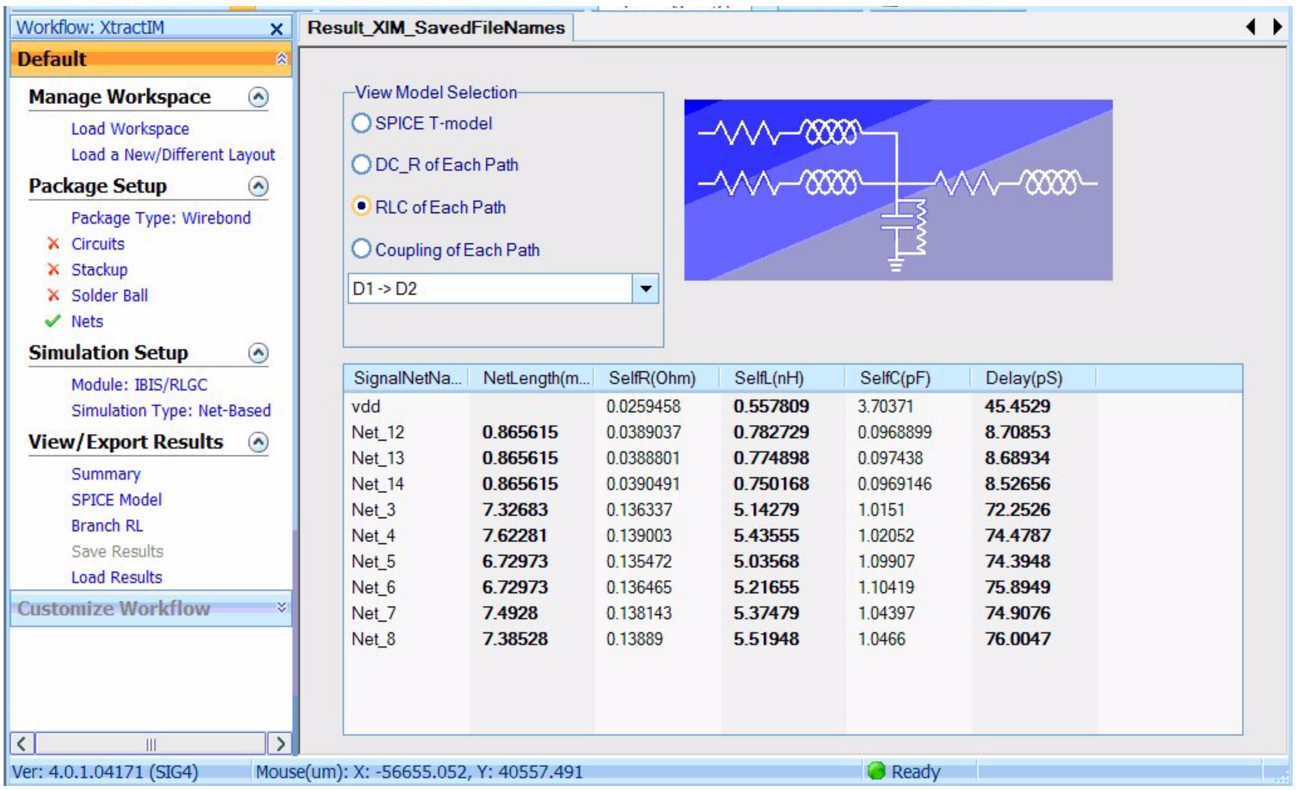
Load in Saved Results and Files

1. To load saved results, select:

   Load Results

2. View the present results or the loaded results.
3. Unload a loaded result by clicking on the **Unload Extractor Result** icon in the toolbar.
4. Use the **Load** and **Unload Result** buttons.

   All output files, including .csv files, are saved in the same directory as the *.spd file.



Lesson Five: Batch Mode Simulation

1. To run a simulation in Batch Mode select:

   Start -> Run

2. Change to the directory where the XtractIM.exe file is located.

Batch Mode Example

If you want to use the project (.spd file) defaulted in the workspace file (.xml file), enter:

xtractIM -b "Full_path_toMy_xml_File*xml filename*"

If you want to use a different project file other than the one in the .xml file, enter:

xtractIM -b "Full_path_toMy_xml_File\xml
filename" "Full_path_toMy_xml_File*new_spd-filename*"

Chapter

5

Optimized Broadband Module: Net-based Simulation for Single-die Single-BGA Packages or Multi-die, Stacked-BGA Packages

This chapter takes you through the steps to use the XtractIM tool Optimized Broadband Module in the net-based simulation of a single-die BGA package. Simulation of multi-die, stacked-BGA packages is the same as single-die, single-BGA packages.

LESSON ONE: SETUP FOR THE SIMULATION

Package and Simulation Setup

1. Load an existing workspace file (.xml file).
2. Open a layout file (.spd file).
3. Select a package type: wirebond or flip-chip.
4. Setup the circuits: select or deselect Die-circuit and Board circuit.
5. Set the Stackup: set parameters for the Bump / Solderball medium layer.
6. Set the Bump data if it is a flip-chip package.
7. Set the Solder Ball data.
8. Select the **nets** for extraction.
9. Select Simulation Module: Optimized Broadband Module.
10. Select output SPICE circuit.
11. Setup the Extraction Frequency band.
12. Save the Workspace and Project file.

Steps 1-8 and Step 11 are exactly the same as ***Preparing for the Simulation*** in ***IBIS/RLGC Module: Net-based Simulation of Single-die Single-BGA Packages***. This chapter describes steps 9-10 in detail.

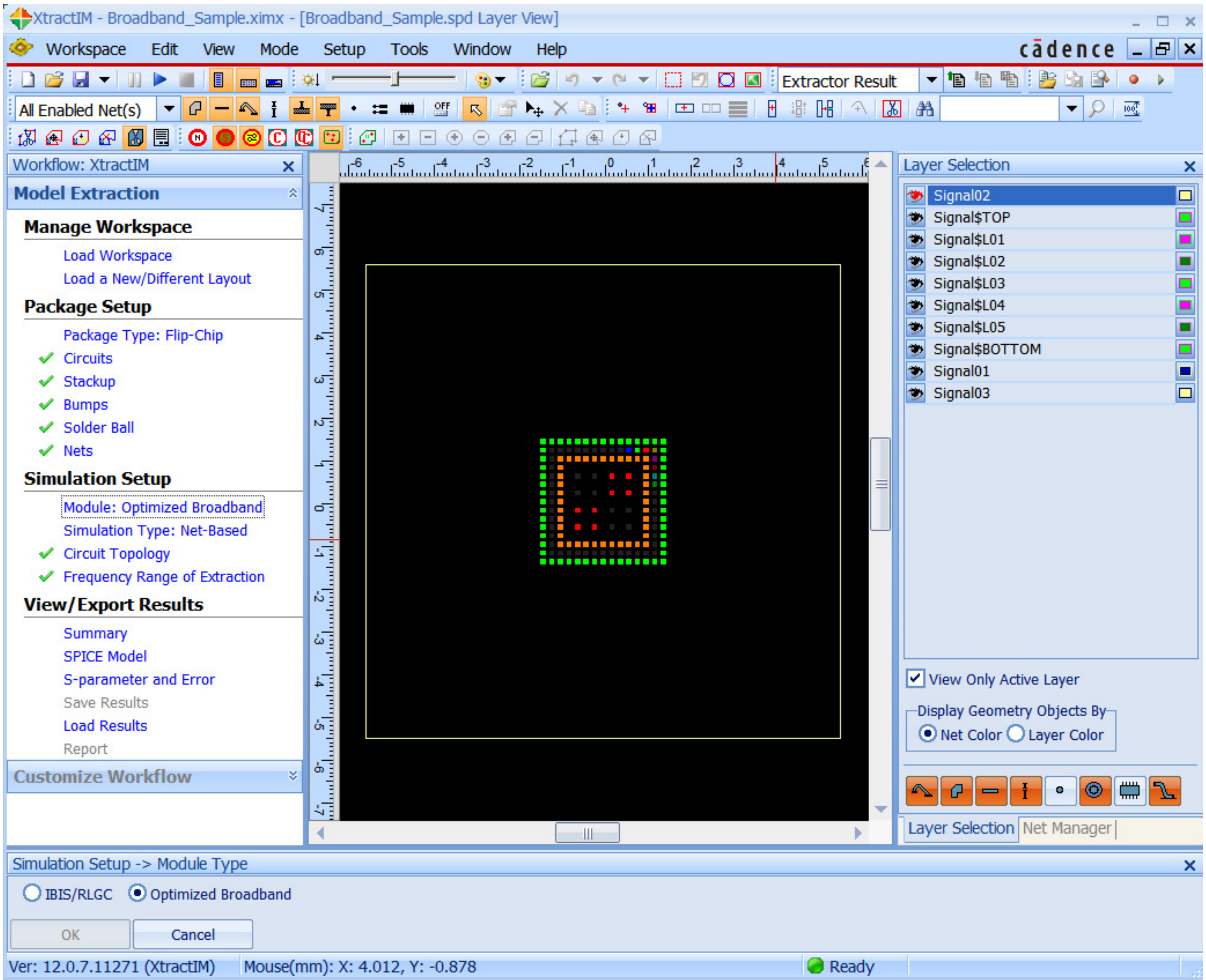
Simulation Setup

1. Select:

   Simulation Setup -> Module

2. Select the IBIS/RLGC **Module**;

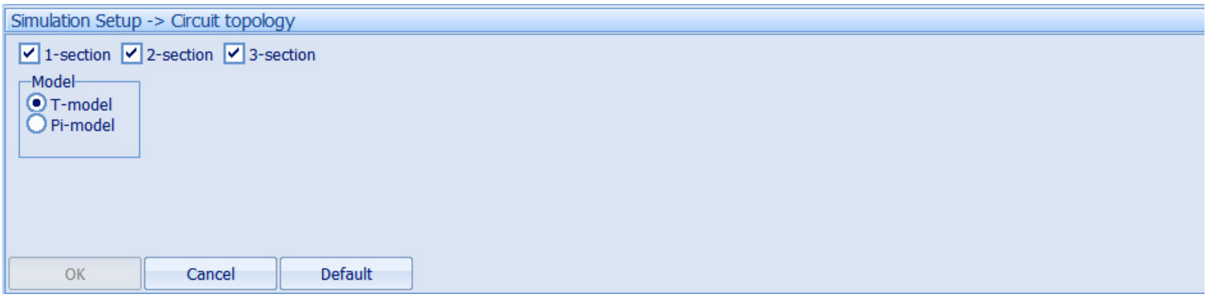
   or select **Module: 6**.



3. In the workflow window click:

   Circuit Topology

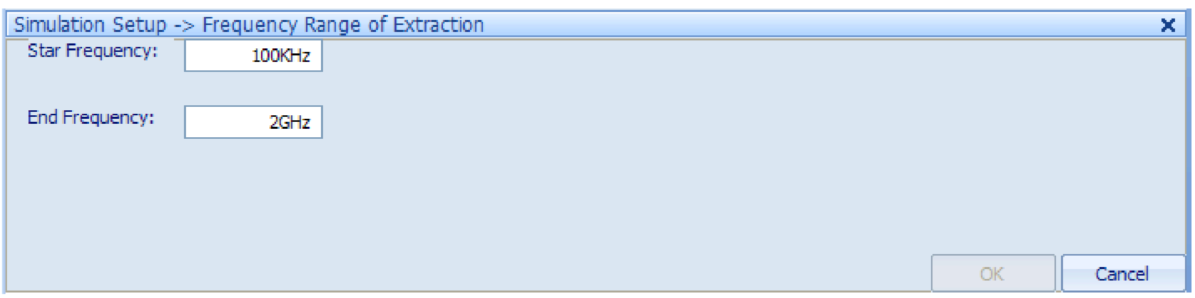
4. Select the circuits to be extracted under the **Circuit Section**.
   - If you do not select a circuit, no RLGC SPICE model is extracted
   - If you select all of the circuits, all three models will be given
   - If you select a single circuit, choose **T-model** or **Pi-model**



5. In the Workflow pane click:

   Frequency Range of Extraction

6. Set the **Frequency Range**.
7. Input the lower frequency and the upper frequency.



Output Frequency Range Options

1. To open the **Output Frequency Range** window, select:

   Tools ->Options -> Edit Options -> Output Frequencies

   The default setting is disabled.

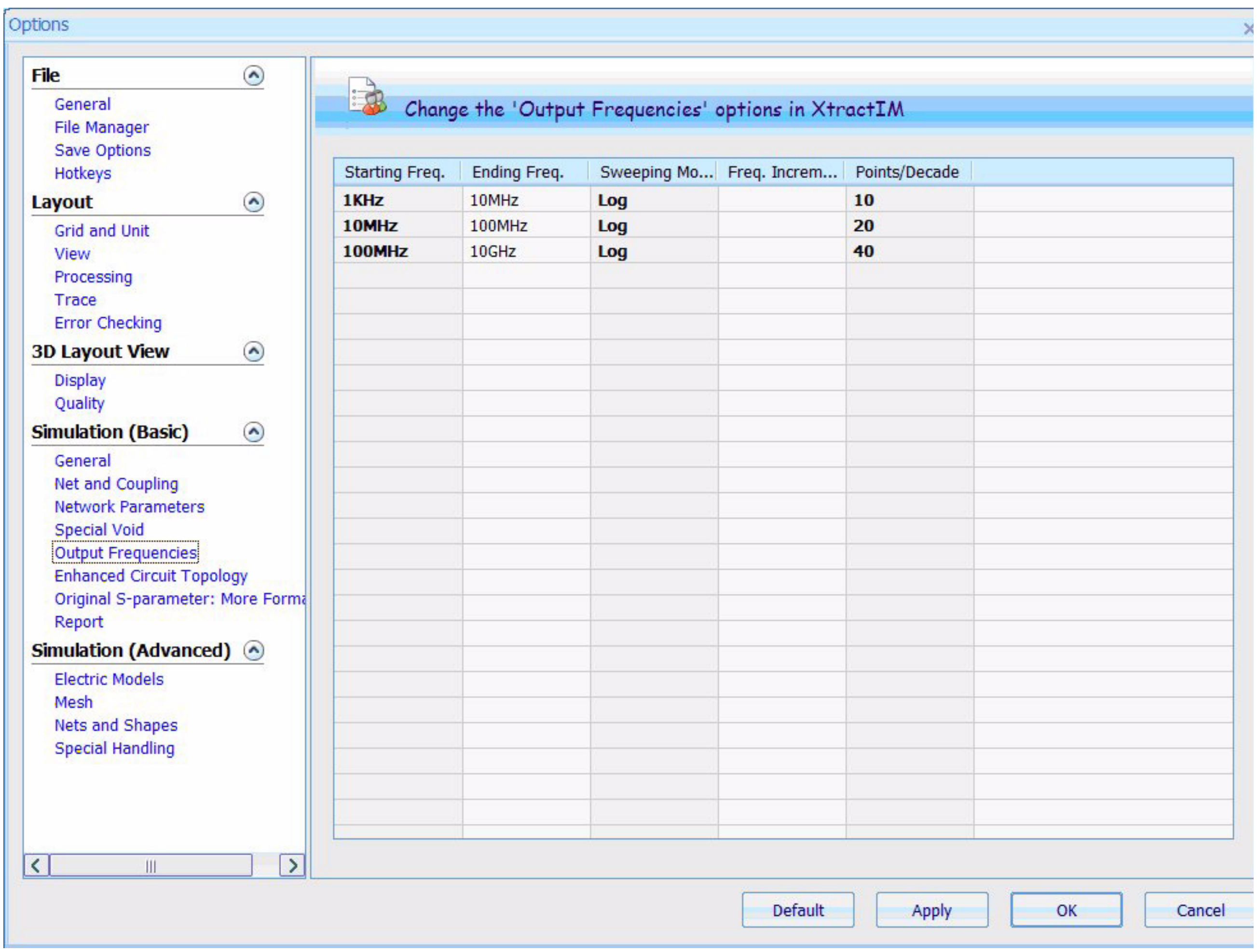
2. Change the sweeping mode and frequency point number in the set Frequency Range window. The default setting for **Enhanced Circuit Topology** is disabled.

   If **Enhanced Circuit Topology** is enabled, the circuit includes more R and L elements to better fit the original S-parameter.

   This option requires 3x - 9x simulation time.

   The default setting is disabled.

3. If the Original S-parameter is enabled. It is saved in Sigrity BNP format and on hard disk in Touchstone format.



Output Original S-parameter with More Format

By default the original S-parameter is saved as a BNP file.

1. Select:

   Tools > Options > Edit Options > Original S-parameter: More Format

2. Select:

   Original S-parameter in Touchstone

   A file with Touchstone format is also saved.

3. Select:

   Original S-parameter in BNP AFS format

   A file with BNP AFS format is also saved.

Lesson Two: Save Work Areas

Layout File

1. Click on the workspace.
2. Click on the **Open Layout File** icon (**green**) to open the project file.
3. To save the layout file under a different name, select:

   Save as

NOTE!	Saving the workspace automatically saves the .spd file. Saving the.spd file does not automatically save the workspace.

Workspace

1. Click on the workspace. The workspace opens.
2. View the Workspace toolbar.
3. Click on the **Save Workspace** icon (**yellow**) in the toolbar.
4. To save the workspace under a different name, select:

   Save as

Lesson Three: Run the Simulation

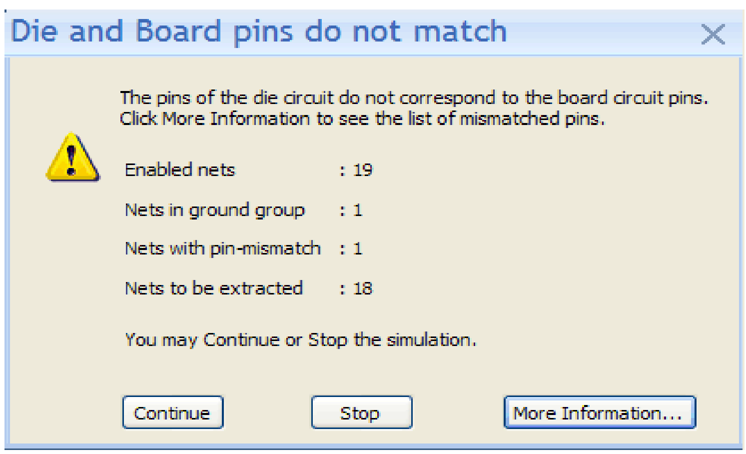
1. Click on the **Play** button  at the top of the window to start the extraction (simulation).
2. The Optimized Broadband **Module** only extracts RLCG for the net which has at least one pin at the Die-side and at least one pin at the board side.

   At the beginning of the simulation, if some nets have Die-Board mis-match, a pop-up window opens.
3. Select the next action.
   - **Continue** — Continue the simulation
   - **More Information** — Examine what nets are mis-matched
   - **Stop** — Cancel the simulation

Investigate Mis-matched Nets

1. The **More Information** window lists all the mis-matched nets.
2. Investigate the mis-matched nets to see whether it is a special design or a defective design.
3. Decide whether or not to proceed with the simulation.
4. Choose **Continue**, **Stop** or **More Information**.

   If 30 seconds pass and the user has not made a choice; then, by default, the simulation continues.



Lesson Four : Observe and Save Simulation Results

XtractIM performs calculations for each net. The calculations include:

- Conductance
- Mutual Capacitance with other Nets
- Mutual Loop Inductance
- Resistance
- Self Capacitance
- Self Loop Inductance

About the S - Parameter and Errors

After the simulation completes, view the S-parameter and errors in the Layer View window

- **Bottom figure** (**S Imaginary**) — Imaginary / phase of the S-parameter element
- **Middle figure** (**S Real**) — Real / amplitude of the S-parameter element
- **Top figure** (**Average error in S**) — Average S-parameter error for each net

View S - Parameter and Errors

1. Drag the black vertical marker to **Net i**. The S_{ii} is shown.
2. Click on net names in the middle column to view the:

   Frequency-averaged S-parameter error of the circuit S-parameter vs. the Original S-Parameter
3. View the port names listed with net names plus port at Die- or Board- side.
4. View the average error for 1-, 2- and 3-section circuit topology.
5. Return to **Net i**.
6. View all sections, single section, or two sections results.
7. View averaged S-parameter errors of a non-optimized or optimized circuit.
8. View the port name in the error column to view any mutual S-parameters.

S-parameter Examples

The error definition of circuit S-parameter vs. original or simulated S-parameter is shown in these examples.

Define an error for each net

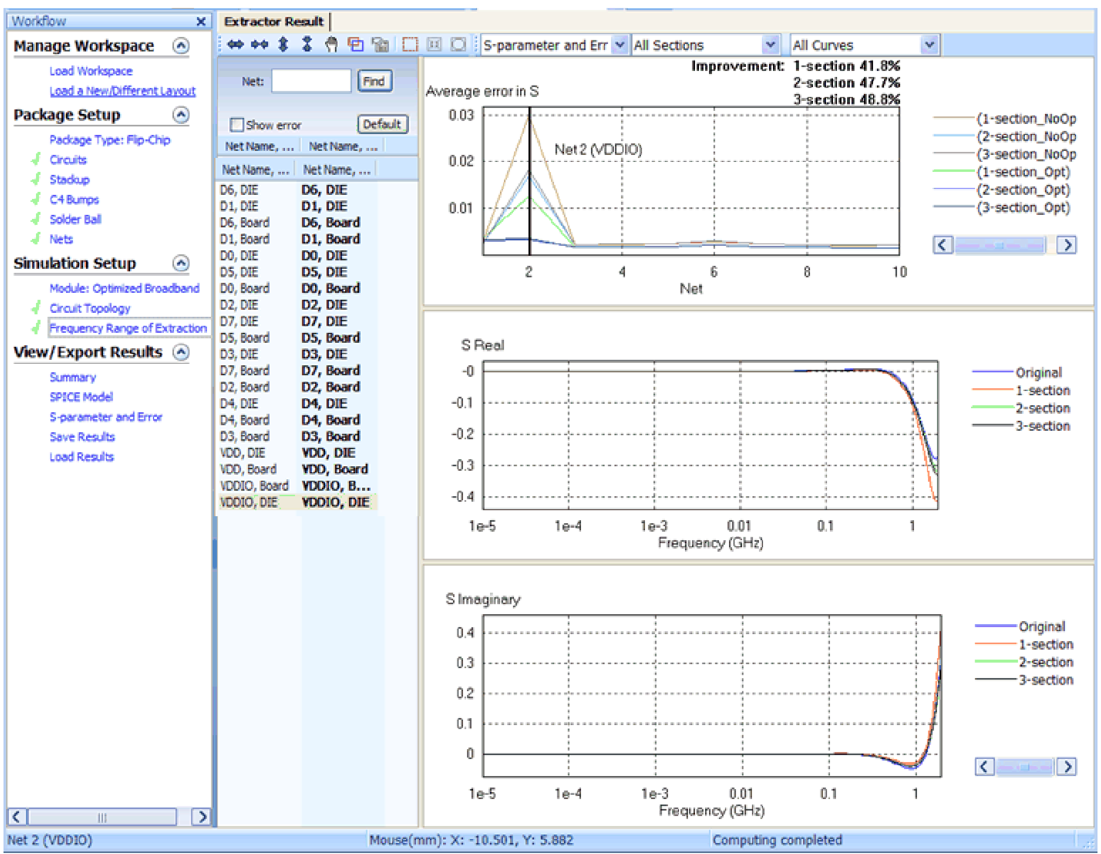
$$Error(net,freq) = \frac{1}{4}\{|S_{11\_ckt} - S_{11\_sim}| + |S_{22\_ckt} - S_{22\_sim}| + |S_{12\_ckt} - S_{12\_sim}| + |S_{21\_ckt} - S_{21\_sim}|\}$$

Net-averaged performance vs. frequency

$$Error1(freq) = \frac{1}{Net\_No}\sum_{Net} Error(net,freq)$$

Freq-averaged performance vs. net

$$Error2(net) = \frac{1}{Freq\_No}\sum_{Freq} Error(net,freq)$$

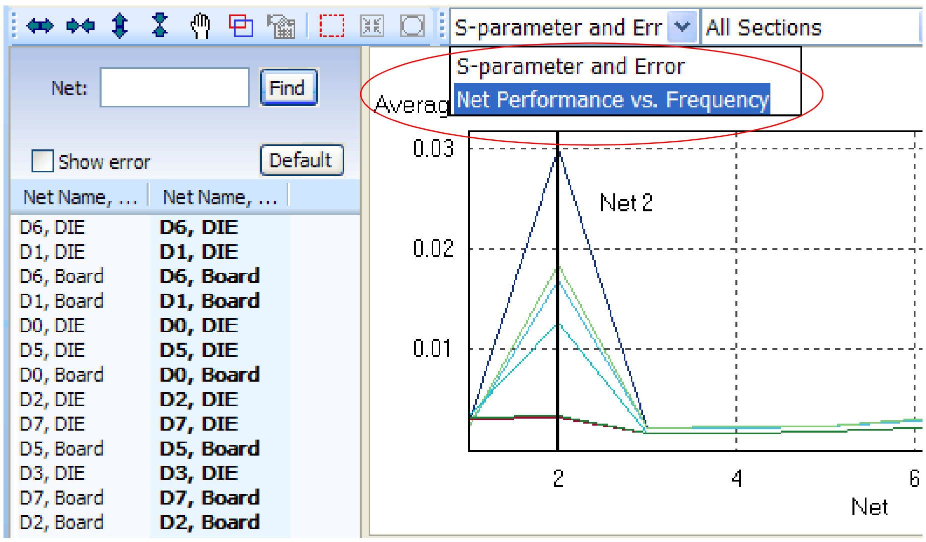


View Net Frequencies

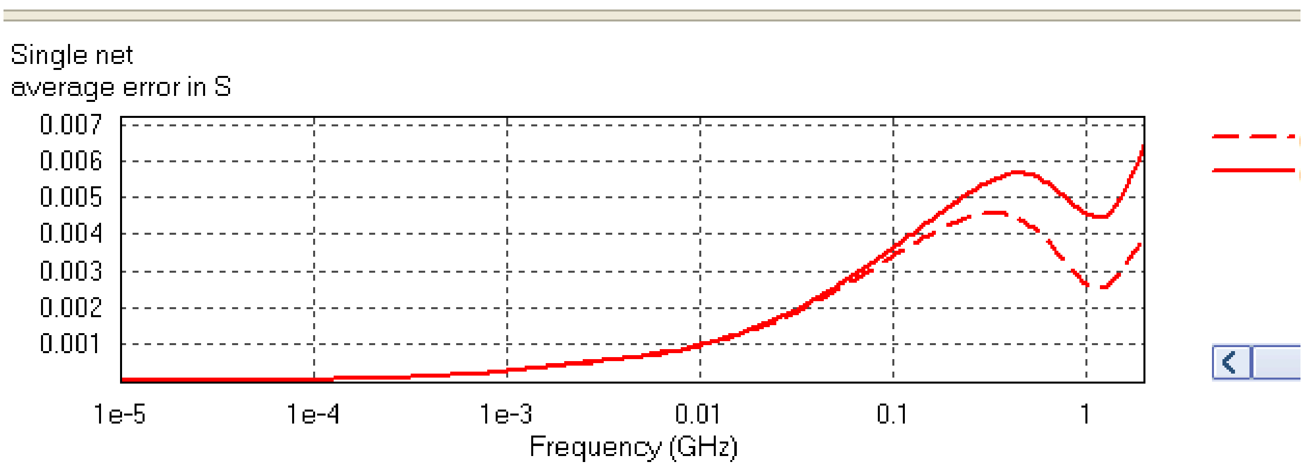
1. Click the **S-parameter** button in the toolbar.
2. In the drop-down menu, select:

   Net Performance vs. Frequency

   The frequency is displayed in the Layer View window.
3. Click on a net name in **Net Name** list.
4. View the Single net average error in **S chart**.

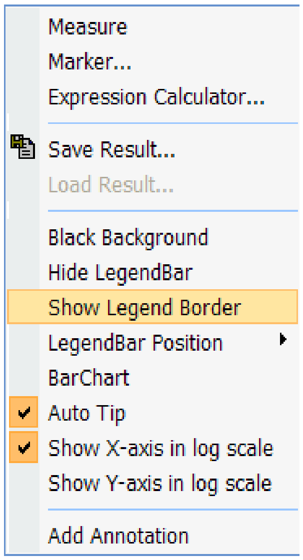


5. Click on each net name in the list.
6. View the Single net average error in S chart for every net.



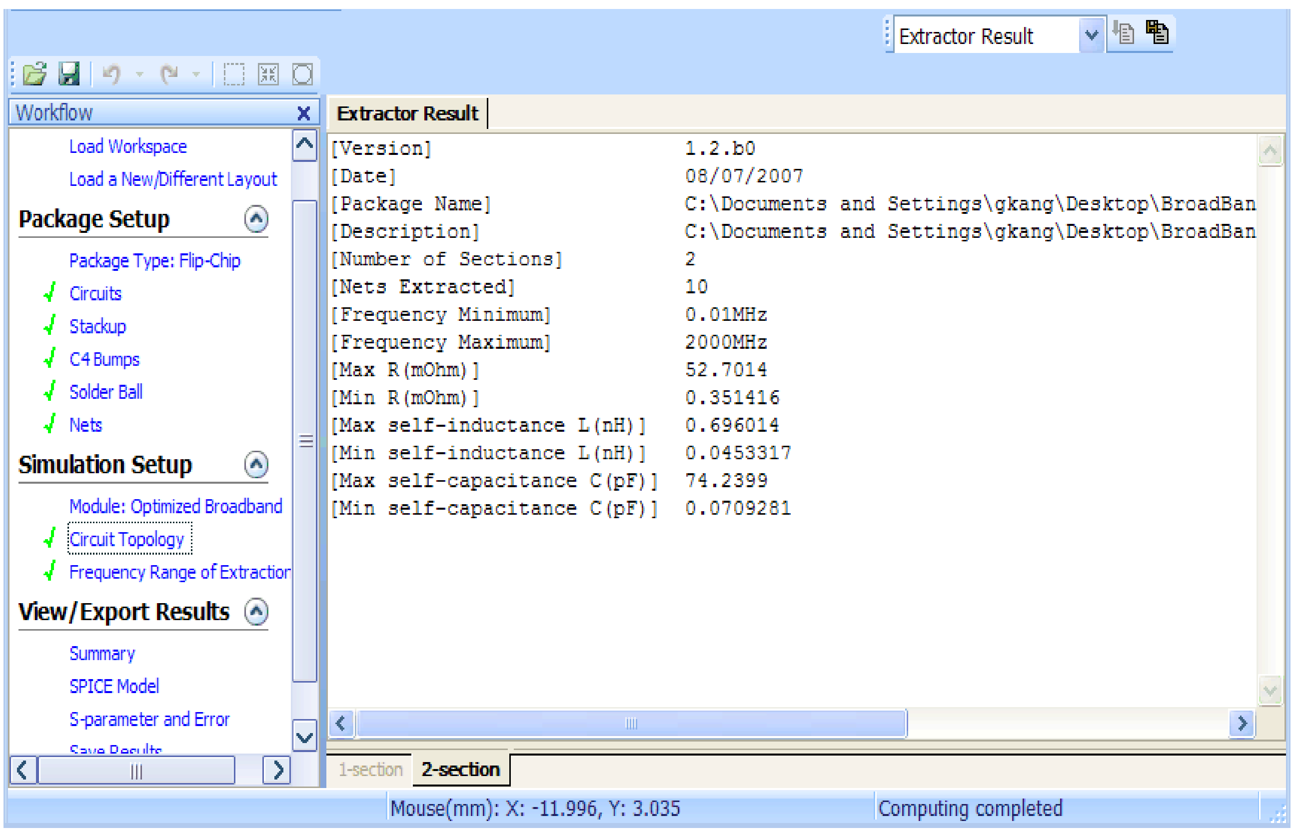
7. Click on a figure. A pop-up menu appears.
8. Click:

Show Legend Border

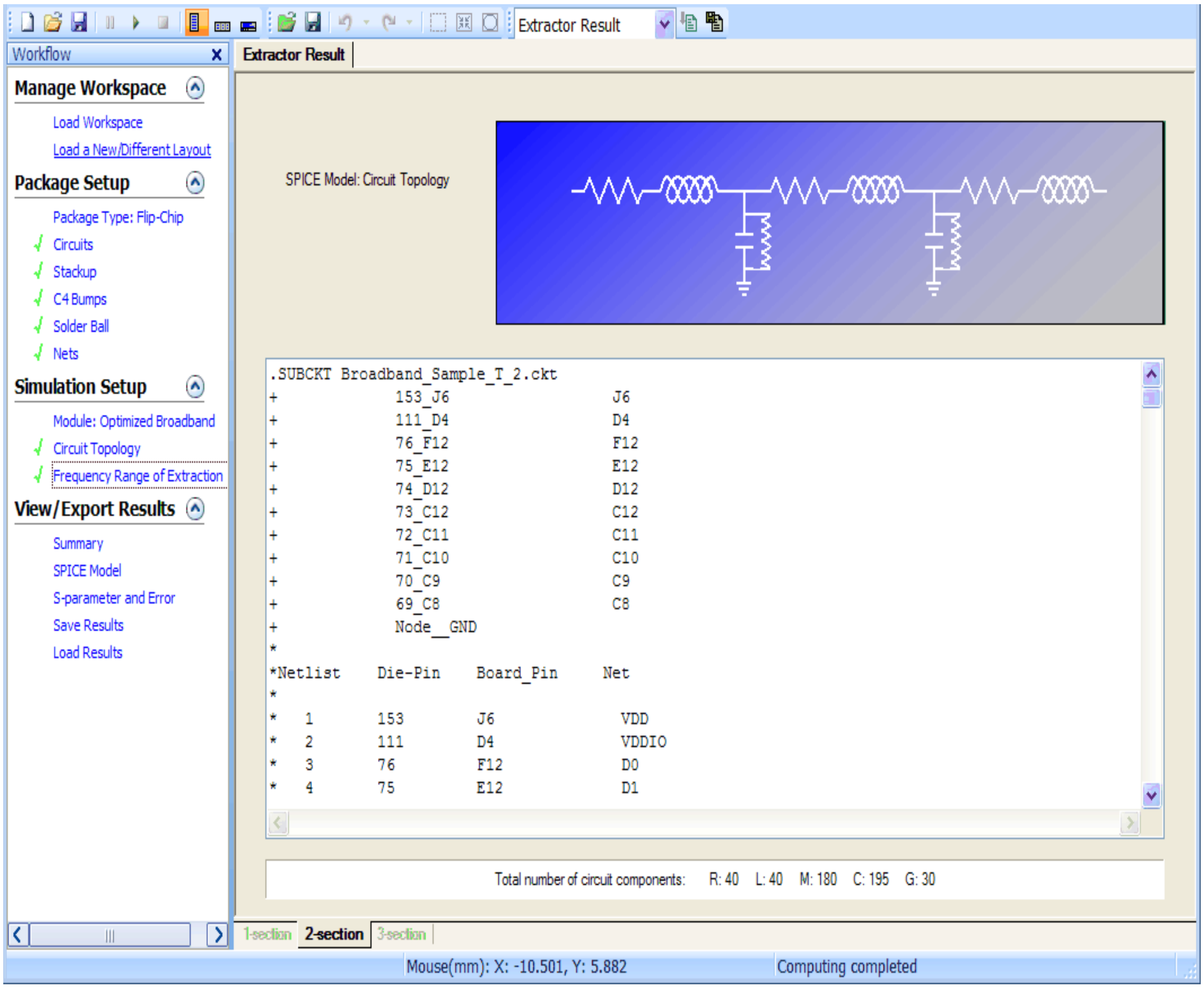


View Summaries and Models

1. Click on **Summary** to bring up the summary and tabulated data for R, L, and C.
2. View the overall results in the top right window.
3. Examine the results for:
   - Frequency Minimum
   - Frequency Maximum
   - Max R
   - Max self-capacitance C
   - Max self-inductance L
   - Min R
   - Min self-capacitance C
   - Min self-inductance L
   - Package Name
4. Click on **SPICE Model** to bring up the display of the SPICE Circuit model.



5. To see the SPICE Model circuit topology click:
   - 1-section
   - 2-section
   - 3-section



Save and Load Results

1. Click **Save Result** in the workflow.The **Save Extractor Result** as window opens.
2. Enter a **name**.
3. Save the **result**.

   The result file:
   - Is saved in a binary file named **result** and **result_spd_file_name.xim**
   - Contains all the files (all the .csv file names)
   - Can be loaded into XtractIM
4. In the Workflow pane click:

Load result

The pre-saved result is loaded.

5. Open the **S-parameter view** file and check the information.
6. Open the **Summary view** file and check the information.

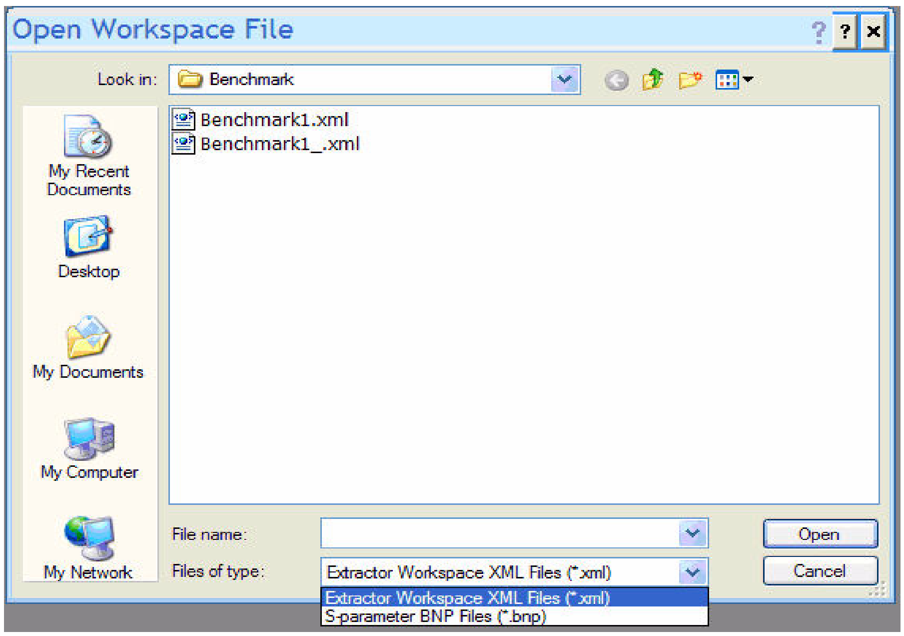
Lesson Five: Broadband Extraction of Existing S-parameter

A typical workflow in interaction mode includes the steps described in ***Load BNP Files and Setup for Simulation***.

Only supported BNP files are created by XtractIM.

Load BNP Files and Setup for Simulation

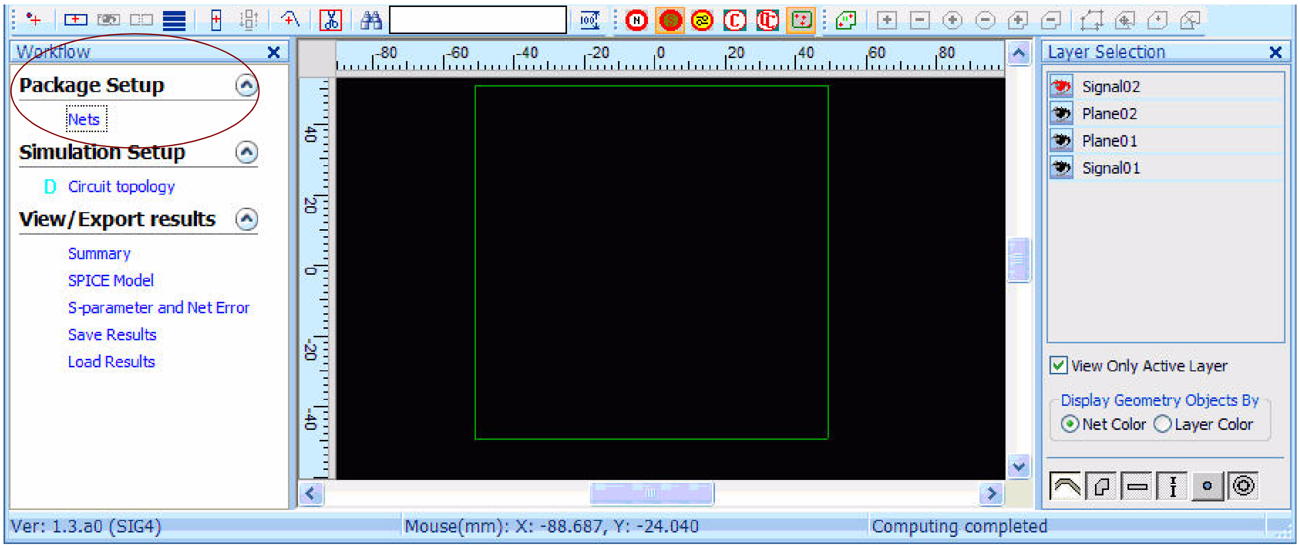
1. Launch **XtractIM**.
2. Open the Workspace file.
3. Select the desired file from the pull-down menu in the **Files of type** field.



4. Select:

   Extractor Workspace XML files

After the S-parameter is loaded the XtractIM window is refreshed.



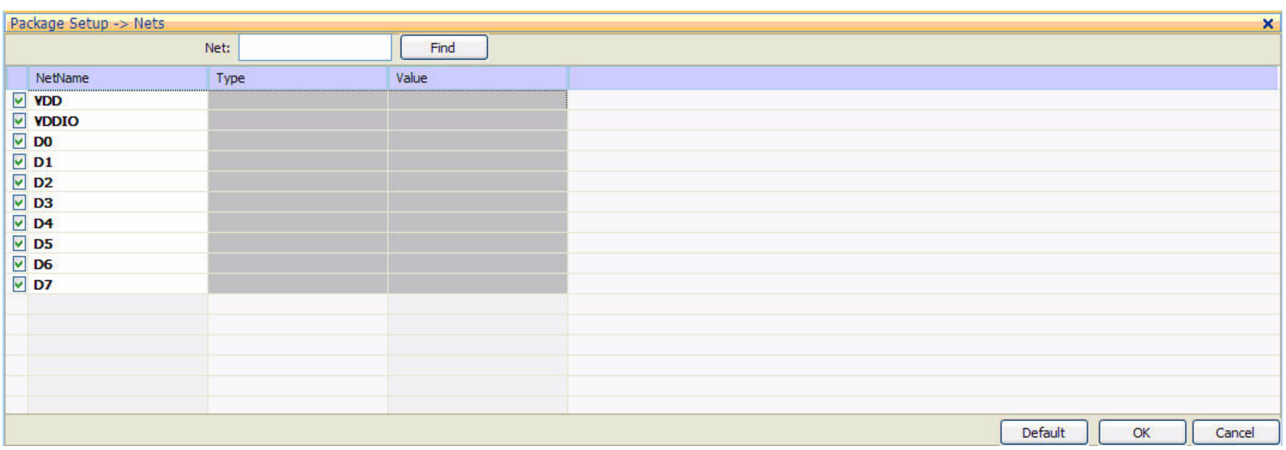
Select Extraction Nets

1. In the Workflow pane, under the **Package Setup** heading, click **Nets**.

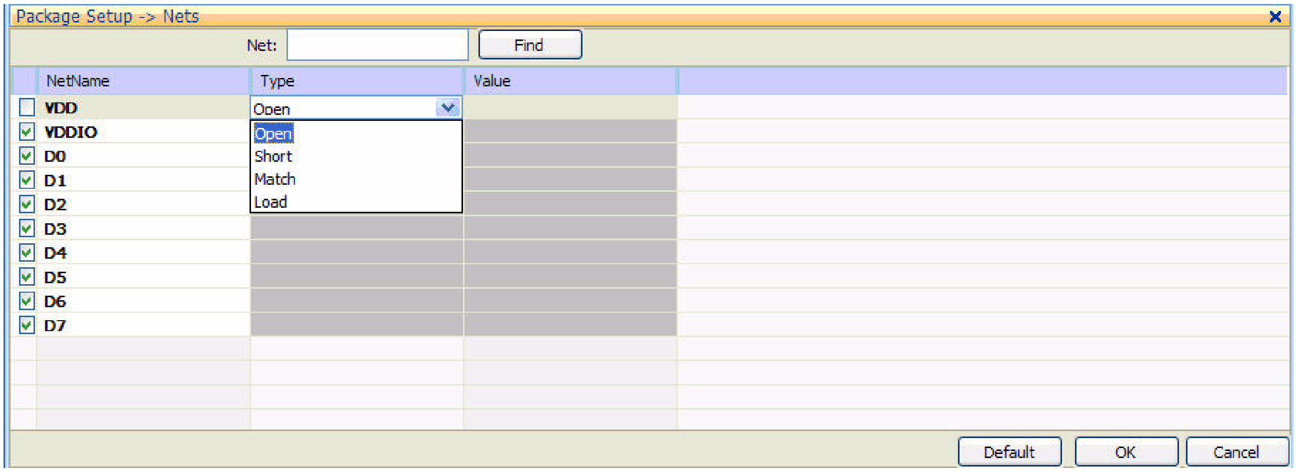
   The Nets window opens.

2. De-select any net that is not extracted.
3. If **VDD** is deselected, the cell in the **Type** column is set on the default **Open**.

   The net is considered open for both ports of this net.



4. Click on **Type** to open the pull-down menu.
5. Select **Load**.
6. In the Value column, enter an actual resistance number (units: Ohm) for a net.


Package Setup -> Nets
Net:
Find
NetName
Type
Value
VDD
Open
VDDIO
Open
Short
Match
Load
D0
D1
D2
D3
D4
D5
D6
D7
Default
OK
Cancel

Select Circuit Topology

1. In the Workflow pane click:

   Circuit Topology

2. To open the pull-down menu click:

   Circuit Section

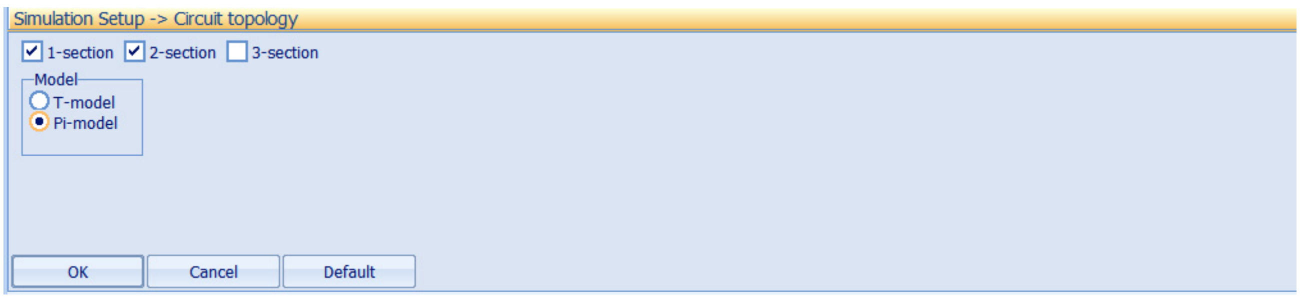
3. Select the circuits to be extracted: **T-model** or **Pi-model**.

If you do not select a circuit, no RLGC SPICE model is extracted.

If you select all of the circuits, all three models are given.

Example

- Pi-model has already been selected for 1-section
- 1- Section and 2- Section models are selected.



Lesson Six: Run a Batch Mode Simulation

1. To run a simulation in Batch Mode, select:

   Start > Run

2. Change to the directory where the XtractIM.exe file is located.

Upon completing the simulation, all output files (including .ckt, .bnp, and .csv files) are saved automatically in the same directory as the *.spd file.

Batch Mode Example

If you want to use the project (.spd file) defaulted in the workspace file (.xml file), enter:

XtractIM -b broadband “Full_path_to_My_xml_File\xml filename”

If you want to use a different project file other than the one in the .xml file, enter:

XtractIM -b -broadband “Full_path_to_My_xml_File\xm filename” “Full_Full_path_to_My_xml_File\new_spd filename”

Chapter

6

RLGC and Optimal Broadband Module: Pin-Based Simulation

This chapter takes you through the steps to use the XtractIM tool in a pin-based simulation of single-die single-BGA or multi-die stacked-BGA packages. The steps for single-die single-BGA RLGC module are strengthened. Those for multi-die stacked-BGA or for Optimized Broadband module are straightforward.

PREPARE FOR THE SIMULATION

Collect this information before you begin the simulation.

- Make sure your files have been translated into SPD format.
- Have the Stackup information ready.
- Have Bump and Solderball diameters, length, heights and conductivity ready.

Lesson One: Simulation Setup

Simulation Overview

A typical workflow in interaction mode is the same as net-base extraction.

A qualified net is the one which has at least one pin in each of the die- and BGA- circuits.

No open circuit exits for these pairs of pins.

Simulation Options

Pin-based extraction has four options in RLGC circuit extraction. The Simulation Setup options are located on the right side of the Editor bar.

- **Option 1** — In DIE or BGA circuit, choose one net as reference. This would lump all pins of the chosen net together as reference in both DIE and BGA side
- **Options 2 & 3** — In DIE or BGA circuit, choose one pin node as reference

  Choose **auto**. XIM automatically selects a node of the net as reference. This node is generally in the center of the circuit

  Choose **manual**. Specify a node as reference.
- **Option 4** — Mesh DIE or BGA circuit. Use one cell element as reference. Any net pins in a same cell are lumped together

Setup Simulation Type

1. Click:

   Simulation Type

2. Select:

   Pin-Based

3. Click:

   Reference Net

   A drop-down menu opens. Qualified nets are displayed.

4. Select one net as the reference net for simulation.
5. Select a circuit from the **Circuit Name Element**.
6. Select one simulation type from the list of four.
   - Use Reference Element
   - Use Reference Net
   - Use Reference Node (auto)
   - Use Reference Node (manual)
7. If you select **Use Reference Net** or **Use Reference Node (auto)**, no further action is needed.

   If you select **Use Reference Node (manual)** or **Use Reference Element**, perform the steps described in the following sections.

8. Repeat steps 5 & 6 for each circuit (DIE and BGA or DIEs and BGAs for multi-die stacked BGA packages).
9. Click **OK** to save all the settings.
10. Click **Cancel** if you want to start over and enter new settings.

Select Reference Node (Manual)

1. Click:

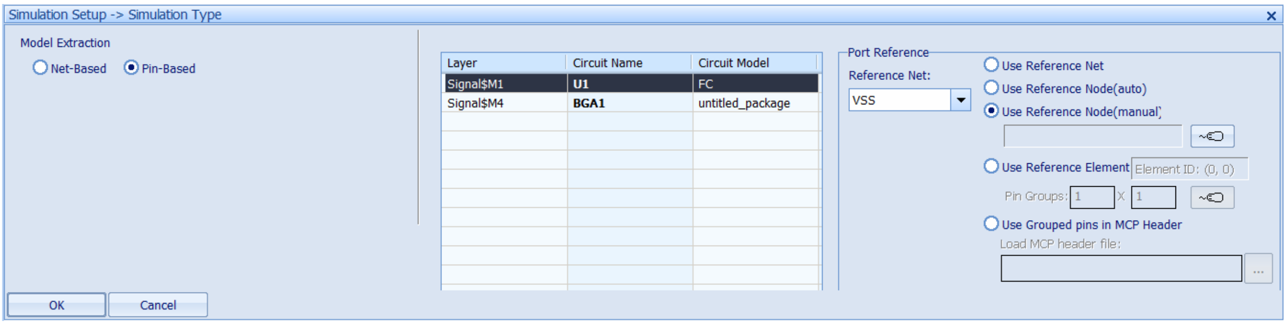
   Simulation Type

2. Select:

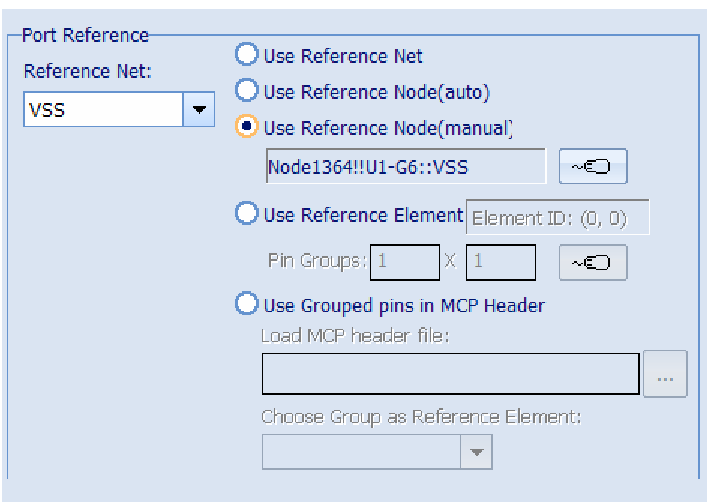
   Pin-Based

3. Click:

   Use Reference Node (manual)



4. Click on the mouse icon.
5. In the **Layout View** window, select a **Reference Net Node**. The Node Name is displayed.



Select Reference Element

1. Select:

   Use Reference Element

2. Choose:

   Pin Groups

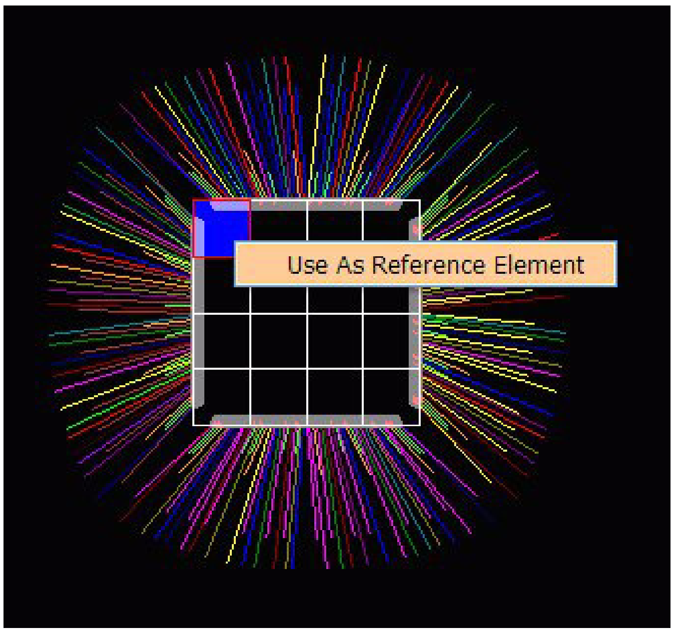
   For example, 1*1, 2*4 or 5*5.

3. Click the mouse icon at the right side of Pin Groups.
4. In the Layer View window right-click to select a cell as the reference element. The selected cell becomes blue.

   A message window appears

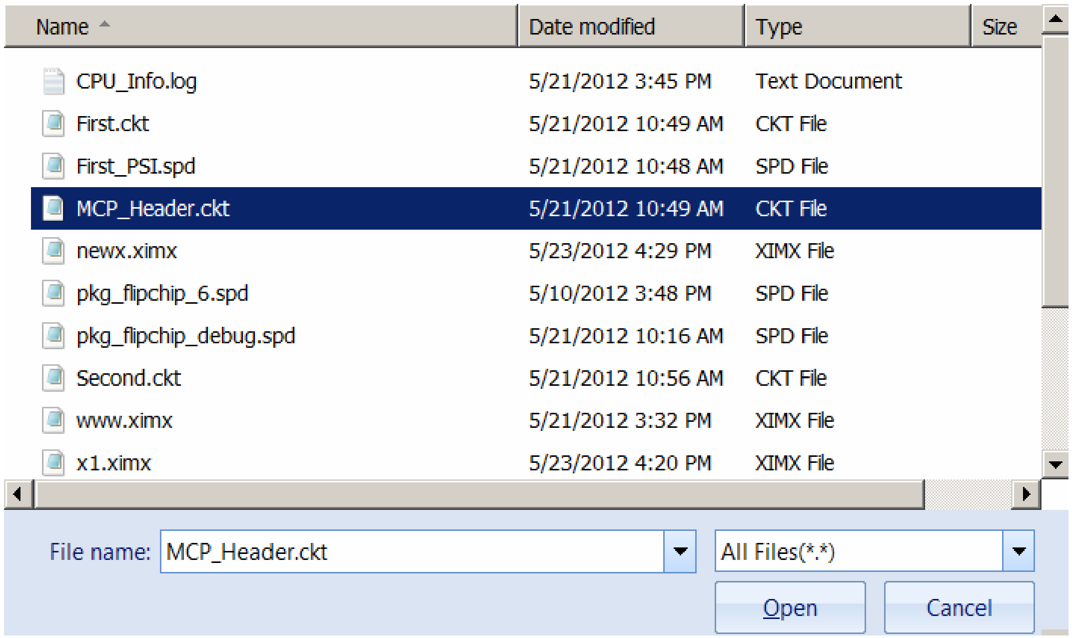
   Use As Reference Element

5. Repeat these steps for other circuits (DIE or BGA)

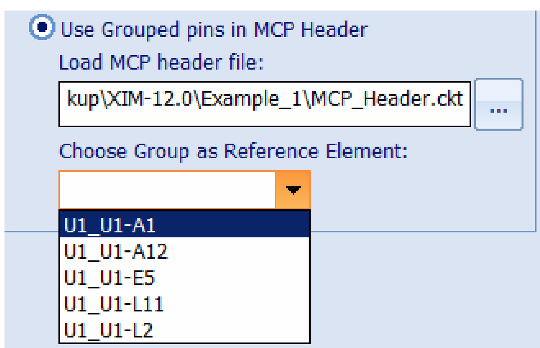


Select User Grouped Pins in MCP Header

1. To load the file with MCP Header, click ... next to the **Load MCP header file** field.



2. Select the MCP Header file (which has the grouped pin information).
3. Click **Open**.
4. In the right side of the **Simulation Setup -> Simulation Type** pane, select a group as **Reference Element** from the drop-down list.



5. Click **OK**.

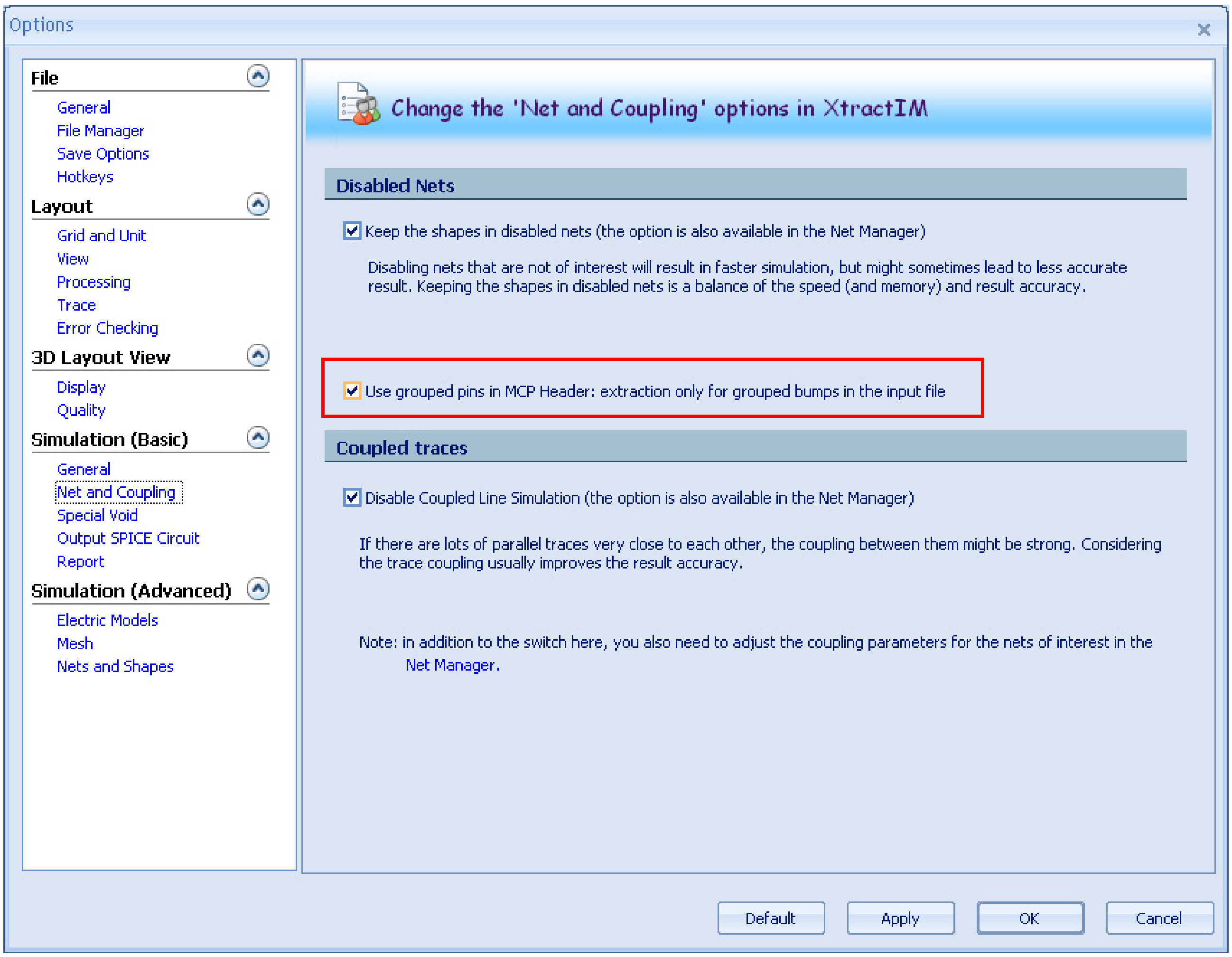
Reference Options

- **Use Reference Net** — Lumps all pin nodes of the Reference Net as port references
- **Use Reference Node (auto)** — Automatically selects a Pin Node of the Reference Net as the port reference; this Node is generally in the center of the circuit
- **Use Reference Node (manual)** — Uses a defined Pin Node for the Reference Net as the Port Reference; we recommend the user choose a Node in the center of the circuit
- **Use Reference Element** — Uses the lump of all the Reference Net Pins in the designed cell as the Port Reference
- **Use Grouped Pins in MCP Header** — Uses the lump of all the reference net pins in the designed group by MCP header as the Port Reference

1) As always, extraction goes to those nets enabled in **Net Manager** only. If a net is included in the MCP Header but is disabled in **Net Manager**, it will not be extracted.

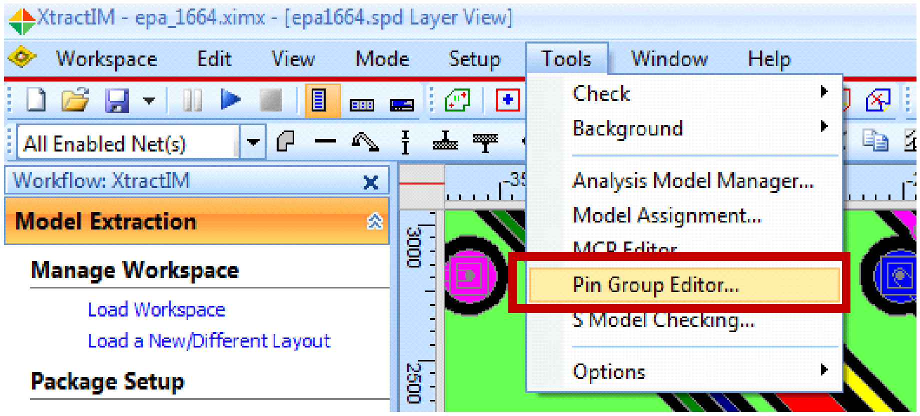
2) If an enabled net is not included in the MCP Header, during extraction, it will be considered as 1-cell by 1-cell lump model extraction.

If the user just wants to extract a model with bumps defined the in the MCP header (i.e. ignore those bumps not in the MCP Header), just select Tools > Options > Edit Options..., and in the **Options** window, click **Net and Couplings**, select the **Use grouped pins in MCP Header: extraction only for grouped bumps in the input file** check box, then click **OK**.



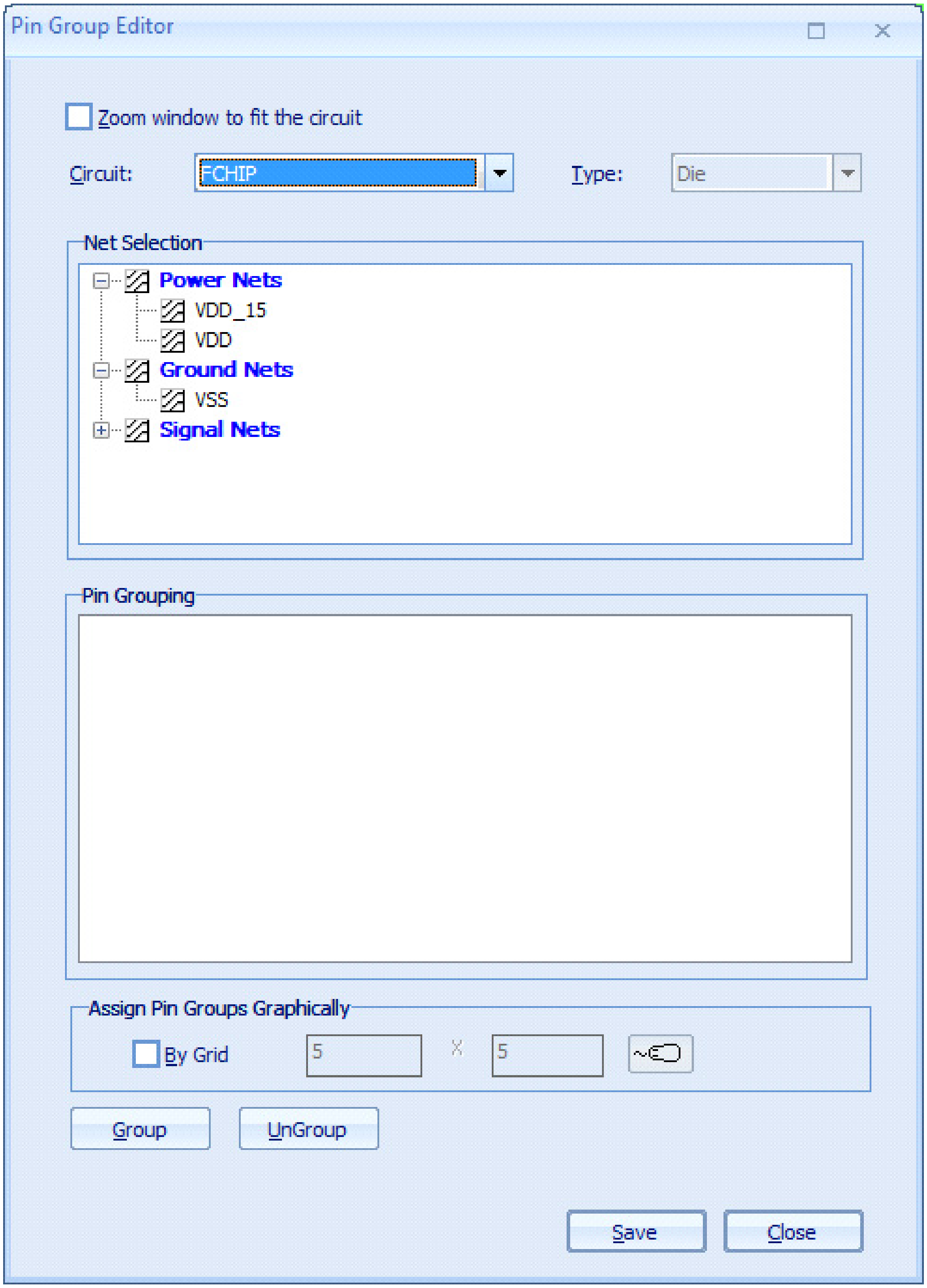
Generate MCP Header File

1. Choose Tools > Pin Group Editor...



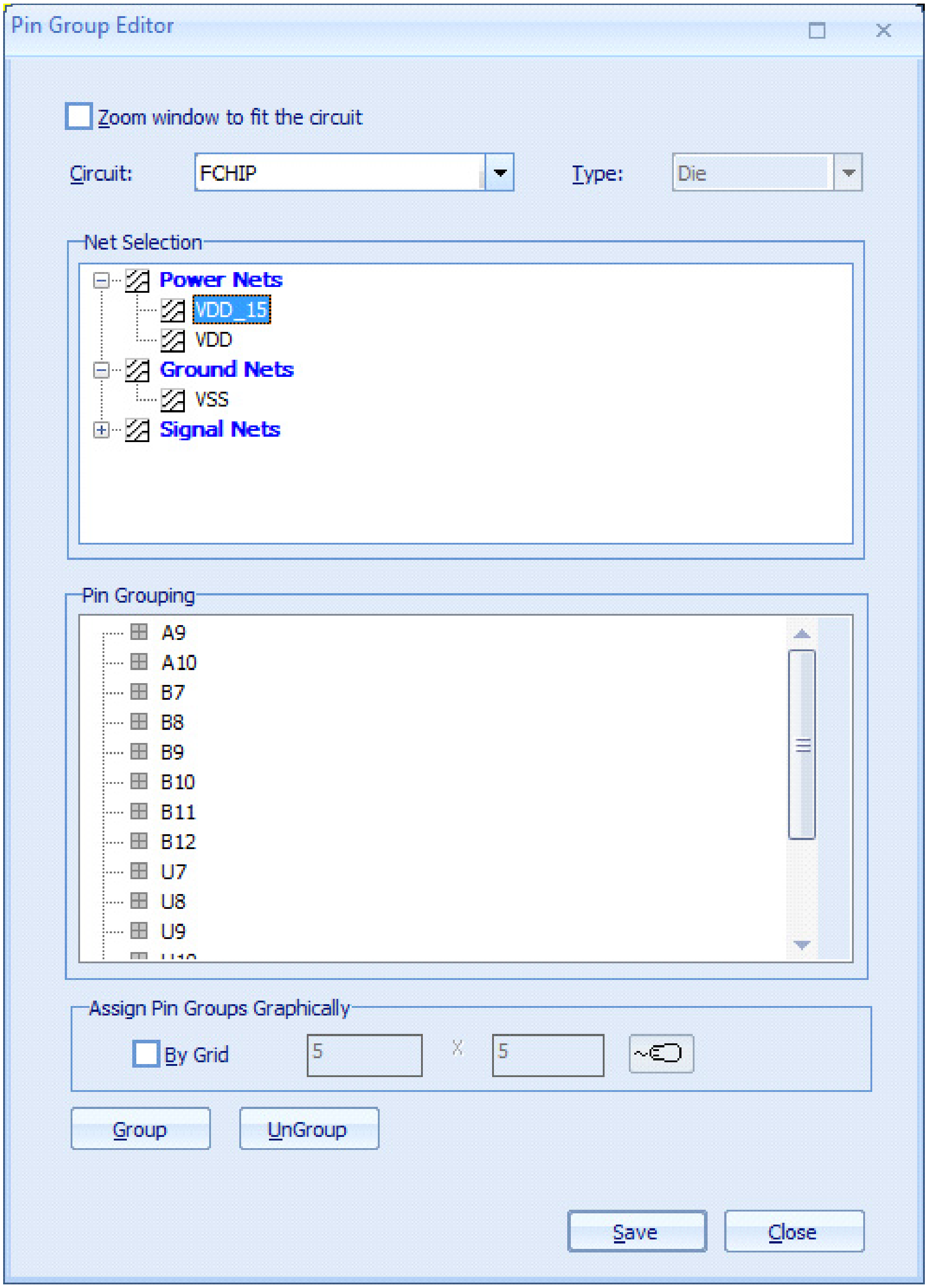
2. Choose an available circuit name from the **Circuit** pull-down list. For example, **FCHIP** in this tutorial.

All of the enabled nets related with this circuit are listed in the **Net Selection** field.

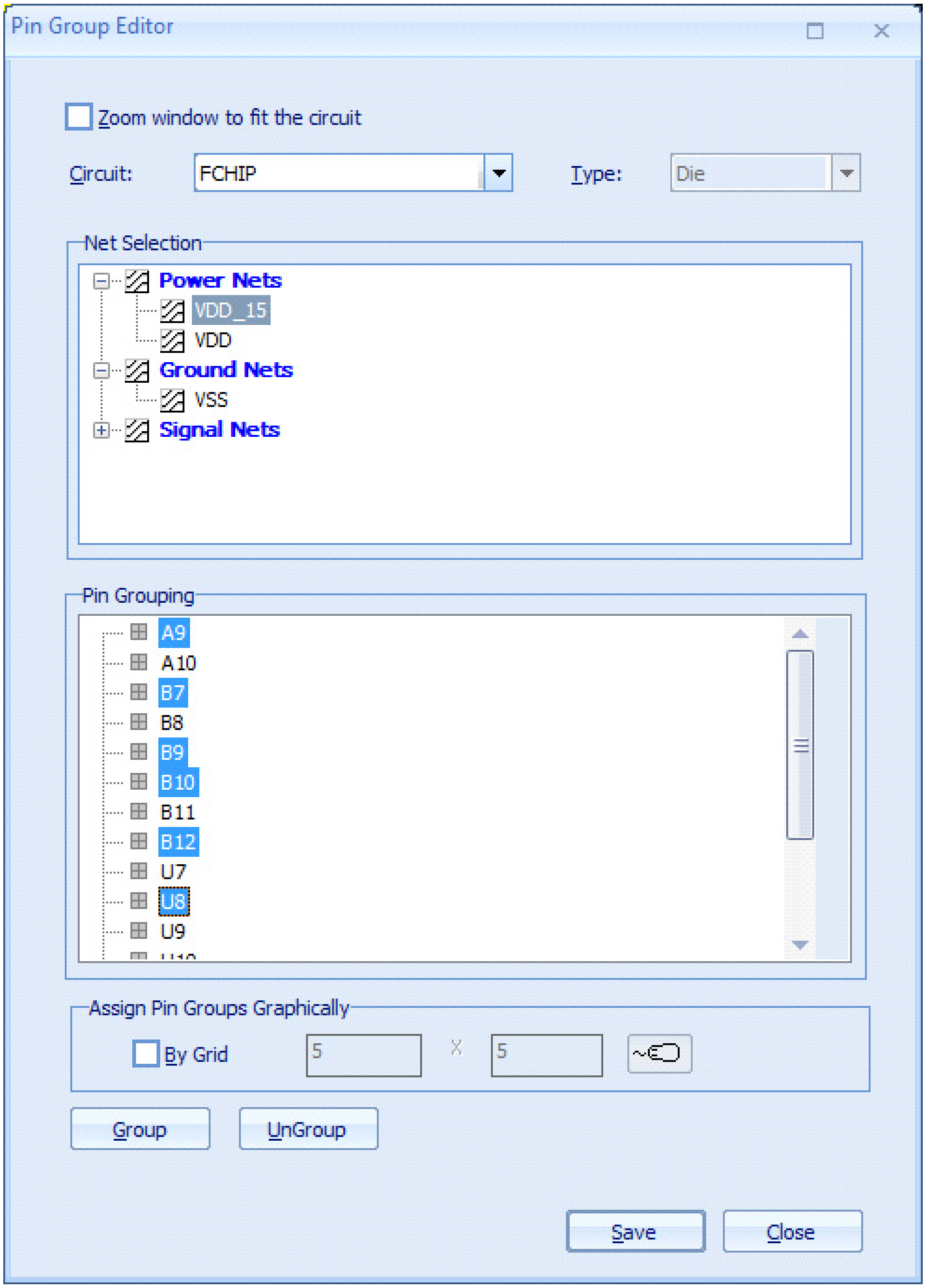


3. Choose a power or ground net to perform pin grouping. For example, **VDD_15** in this tutorial.

The corresponding pins for the selected net are listed in the **Pin Grouping** field.

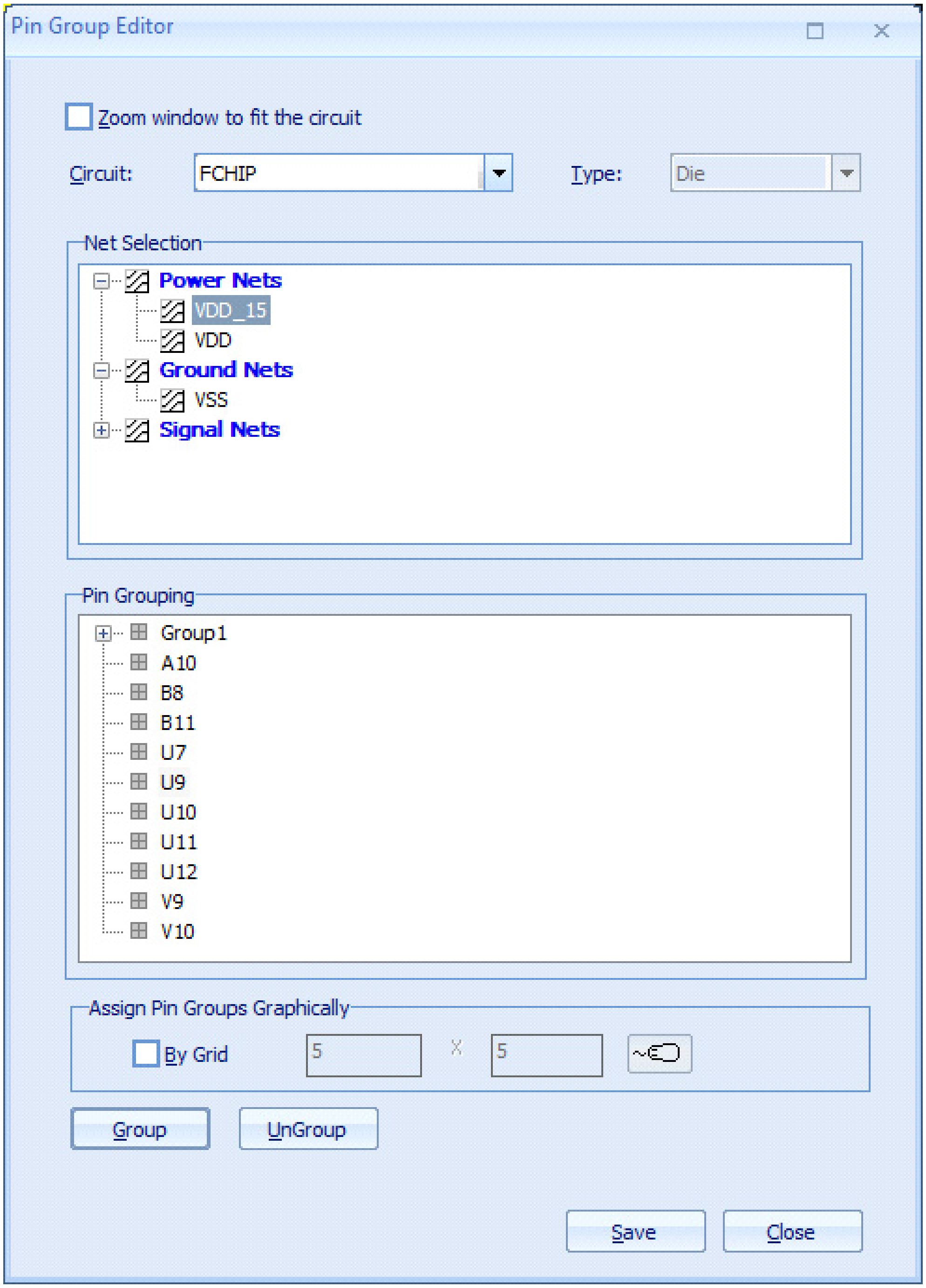


4. Choose pins that you want to add in a group.

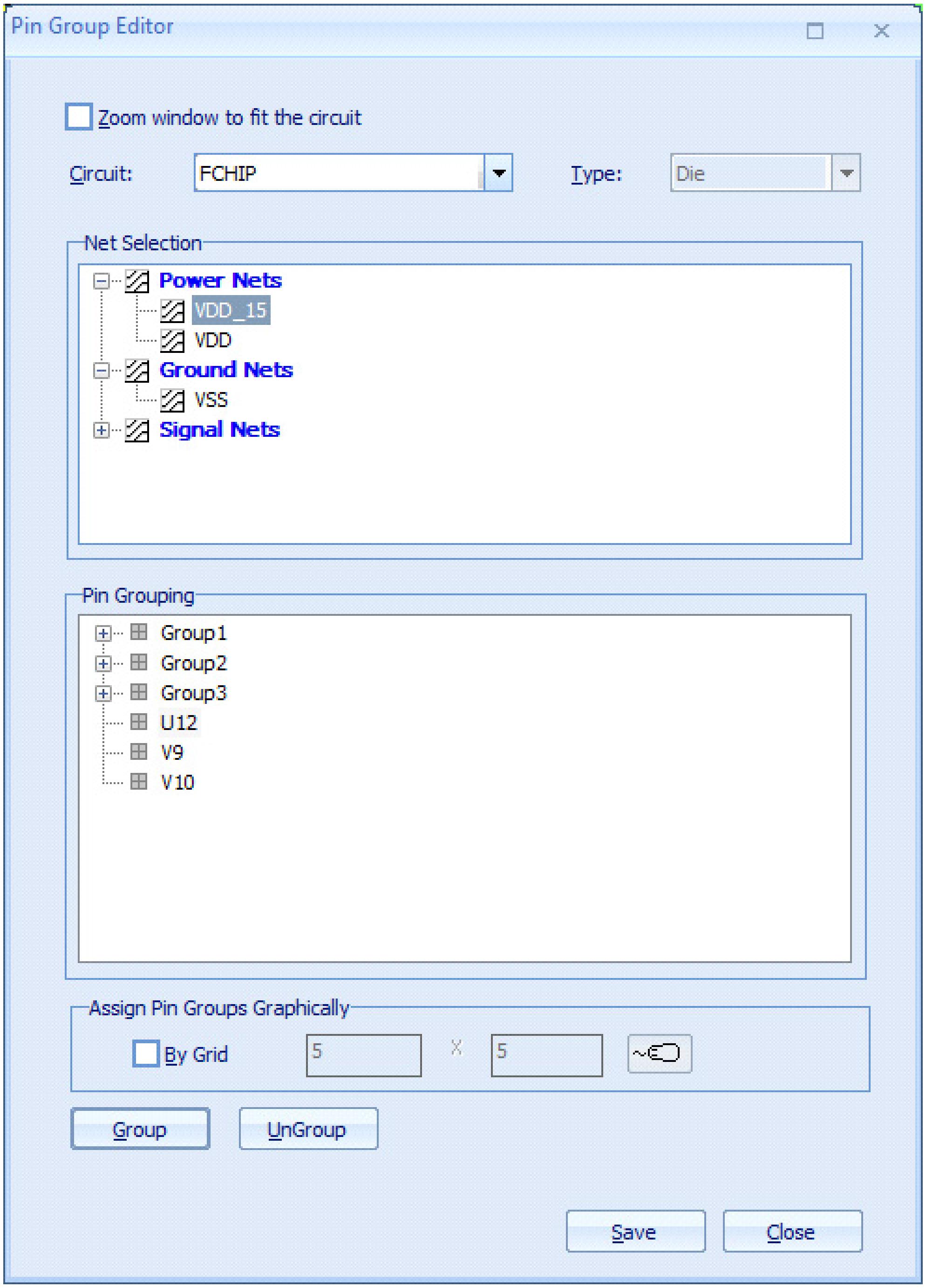


5. Click the **Group** button.

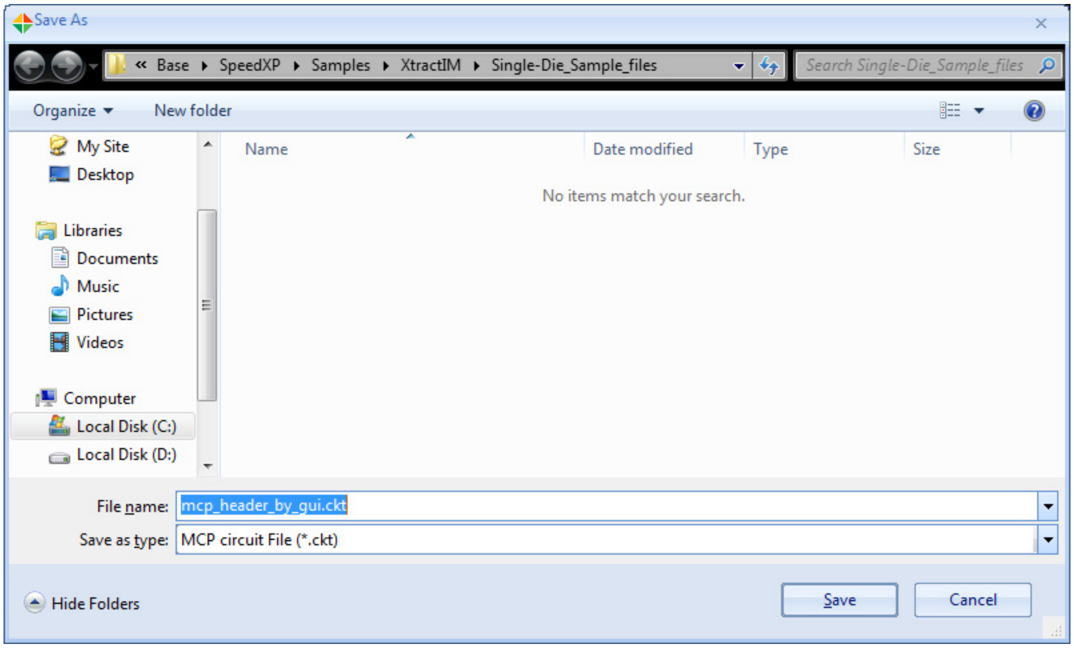
A group is generated as **Group1**.



6. Repeat **Step 4** and **5** to generate multiple groups for a net.



7. Once the pin groupings have been done for all circuits and nets, click the **Save** button to generate a MCP header file, for example **mcp_header_by_gui.ckt**.



The .ckt file generated in the above steps can be used in the ***Select User Grouped Pins in MCP Header*** secion.

Setup Extraction Frequency

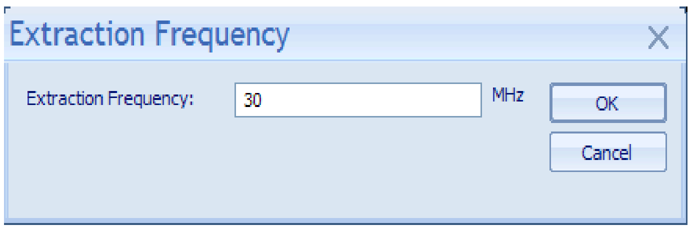
1. Select:

   Setup > Extraction Frequency

   A pop-up window opens.

2. Update the data in the window.

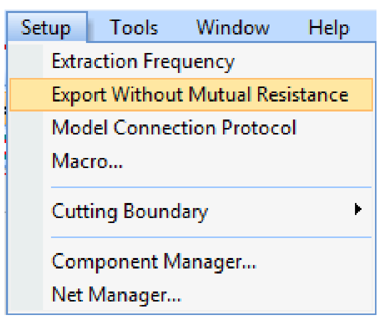
3. Change the default value. The default value is 30MHz.



4. To ignore mutual resistance in RLGC circuits, select:

   Setup > Export Without Mutual Resistance

   The output circuit does not include the H-elements.



Lesson Two: Save Work Areas

Layout File

1. Click on the Workspace.
2. Click on the **Open Layout File** icon (**green**) to open the project file.
3. Use **Save As** to save the layout file under a different name.

NOTE!	Saving the workspace automatically saves the .spd file. Saving the.spd file does not automatically save the workspace.

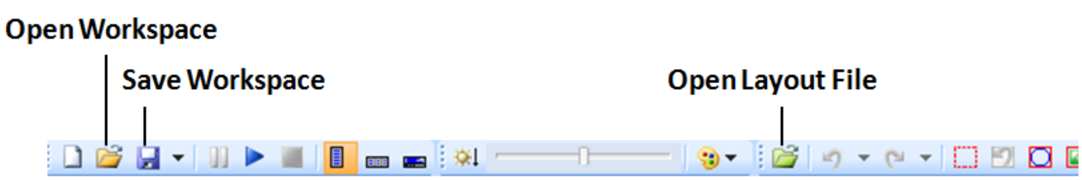
Workspace

1. Click on the Workspace. The Workspace pane opens.
2. View the Workspace toolbar.
3. In the toolbar click

   Save Workspace

4. To save the Workspace under a different name, use:

   Save As



Lesson Three: Run the Simulation

1. Click on the **Play** button  at the top of the window to start the extraction (simulation).
2. XtractIM only extracts RLCG for a net which has at least one pin at the Die side and at least one pin at the board side.
3. At the beginning of the simulation, if some nets have Die-Board mis-match, a pop-up window opens. Select the next action.
   - **Continue** — Continue the simulation
   - **More Information** — Examine what nets are mis-matched
   - **Stop** — Cancel the simulation

Investigate Mis-matched Nets

The **More Information** window lists all the mis-matched nets.

Investigate the mis-matched nets to see whether it is a special design or a defective design. Decide whether or not to proceed with the simulation.

Choose **Continue**, **Stop** or **More Information**. If 30 seconds pass and the user has not made a choice; then, by default, the simulation continues.

Lesson Four: Fly-Wirebond Package Model Extraction

Introduction

For some wirebond package designs, there are wire connections only among die pads for power and ground nets. These die pads don't have connection to bond fingers or to package pins. Such kind of package is usually named as package with fly-wirebond or jump-wirebond inside. The fly-wirebond is typically used to reduce power IR drop noise.

In this lesson, it is introduced how to get a package model with fly-wirebonds in XtractIM.

Workflow

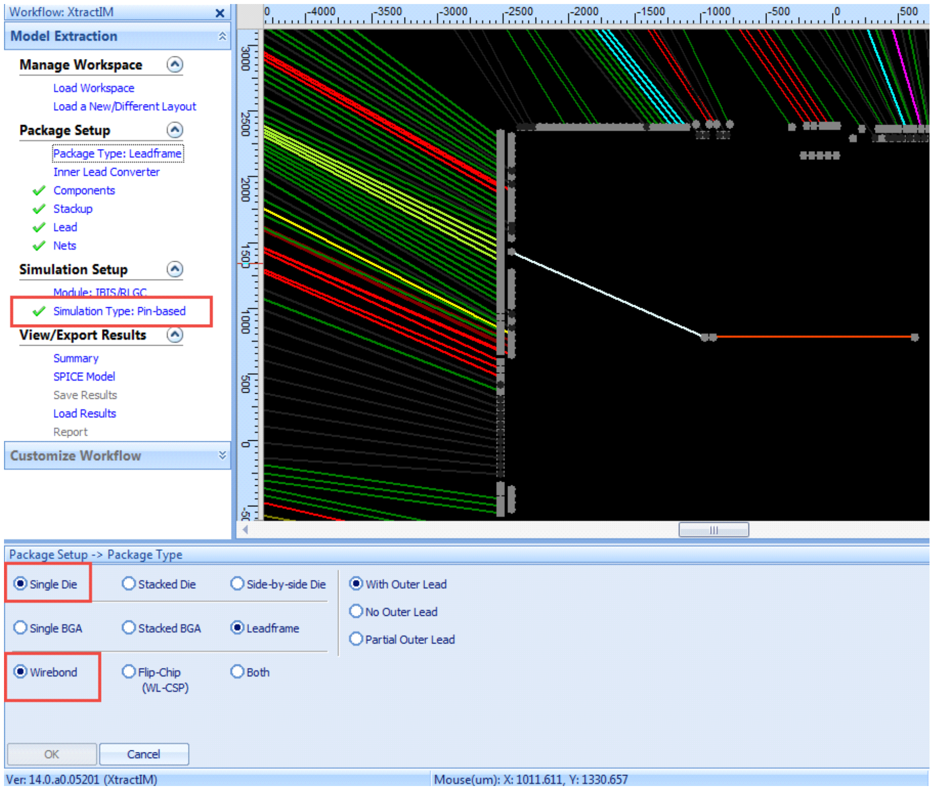
In general, the fly-wirebonds only happen on the same die in a package. It is only supported in single-die/single-package designs flow. The package type can be either BGA or lead-frame.

1. Package Setup:
    a. Select **Single Die**.
    b. Select **Single BGA** or **Leadframe**.
    c. Select **Wirebond**.
2. Simulation Setup:

    The fly-wirebond is only supported in the pin-based mode.

    a. Select simulation type: **Pin-Based**.

No specific setting is needed.

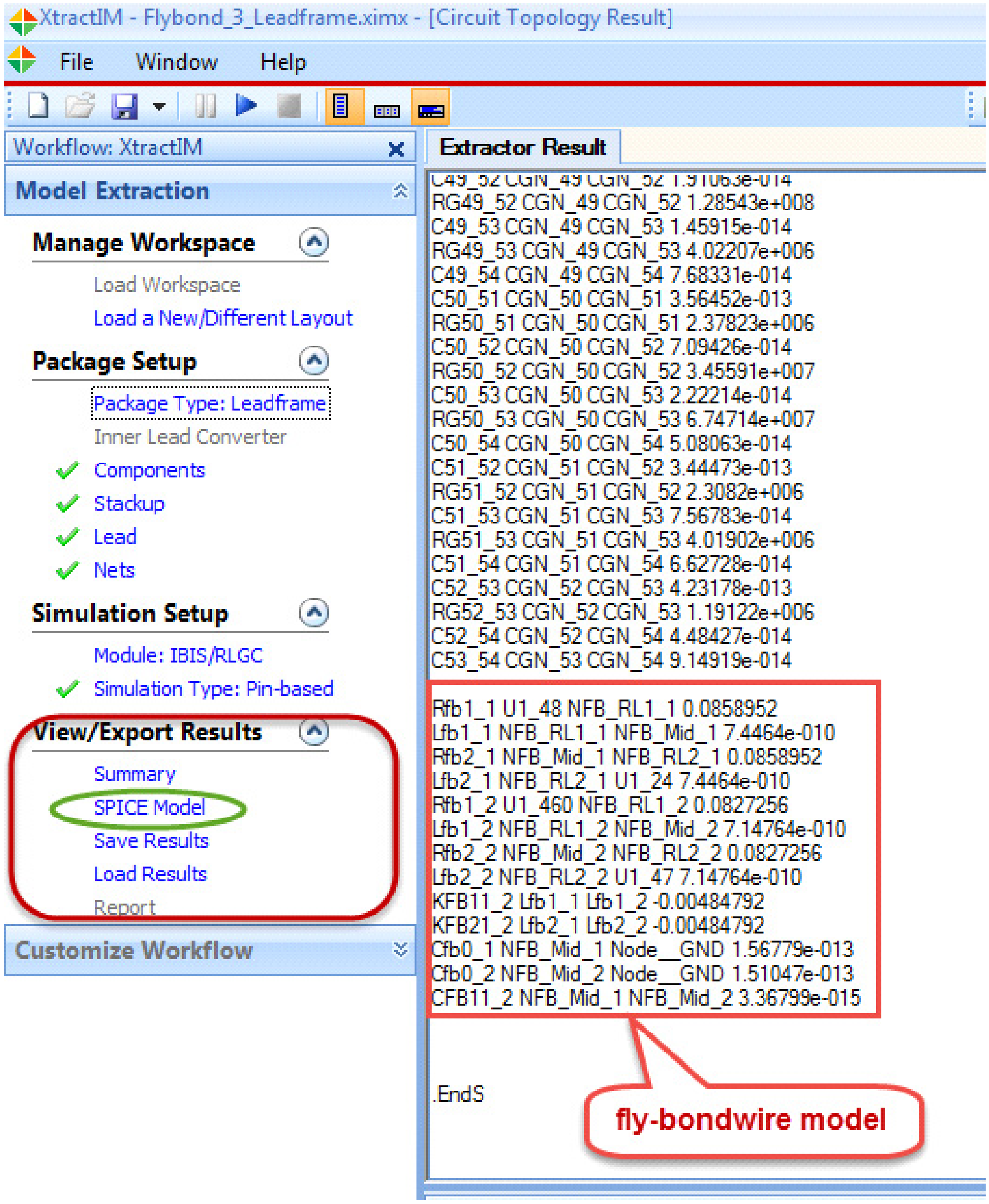


Result

The fly-wirebonds are modeled separately by the tool.

The fly-wirebond model is wrapped into SPICE subcircuit model named

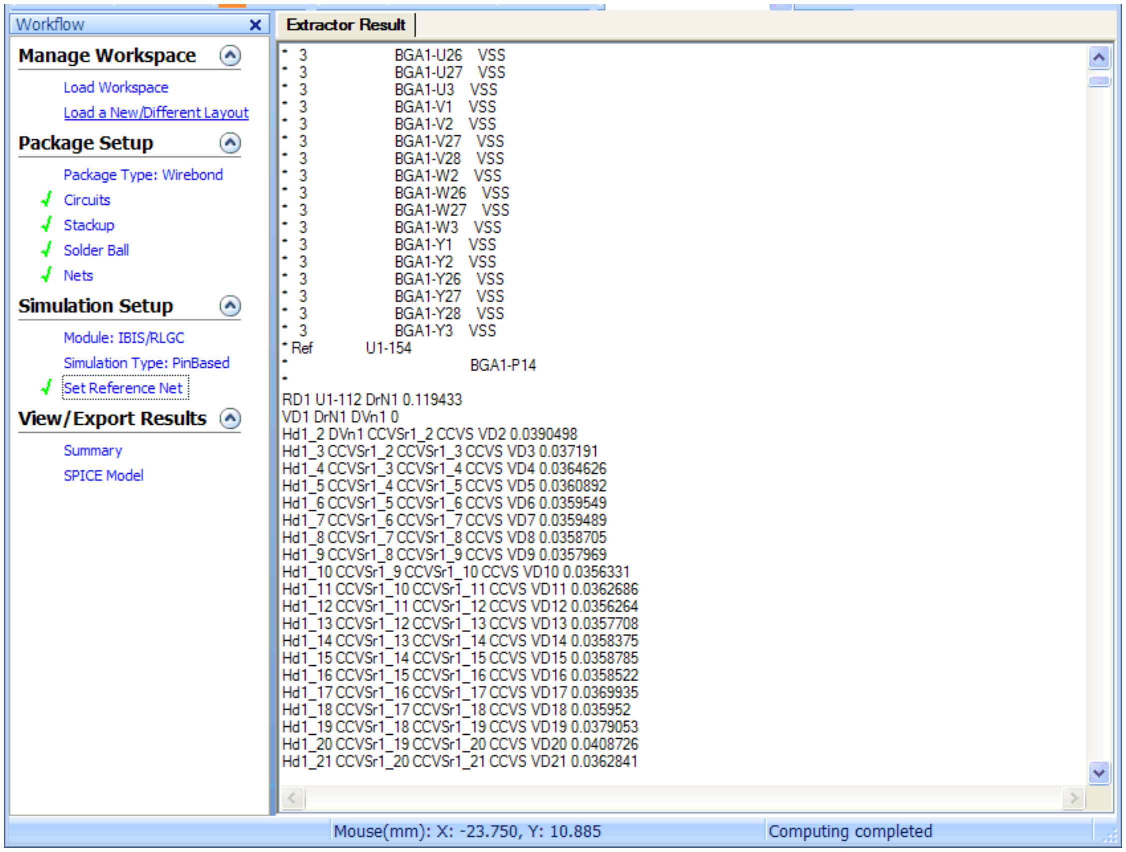
jobname_PinBaseSPICE.ckt as shown below.



VIEW EXPORT RESULTS

The **View / Exports Results** options are displayed on the left side of the window.

1. Click on **Summary**.
2. View the maximum and minimum R, L, C, and other information in the Summary.
3. Click on **SPICE Model**. The SPICE Model name *.ckt is loaded.
4. View the SPICE Model file.


Workflow
Extractor Result
Manage Workspace
Load Workspace
Load a New/Different Layout
Package Setup
Package Type: Wirebond
Circuits
Stackup
Solder Ball
Nets
Simulation Setup
Module: IBIS/RLGC
Simulation Type: PinBased
Set Reference Net
View/Export Results
Summary
SPICE Model
* 3 BGA1-U26 VSS
* 3 BGA1-U27 VSS
* 3 BGA1-U3 VSS
* 3 BGA1-V1 VSS
* 3 BGA1-V2 VSS
* 3 BGA1-V27 VSS
* 3 BGA1-V28 VSS
* 3 BGA1-W2 VSS
* 3 BGA1-W26 VSS
* 3 BGA1-W27 VSS
* 3 BGA1-W3 VSS
* 3 BGA1-Y1 VSS
* 3 BGA1-Y2 VSS
* 3 BGA1-Y26 VSS
* 3 BGA1-Y27 VSS
* 3 BGA1-Y28 VSS
* 3 BGA1-Y3 VSS
* Ref U1-154
* BGA1-P14
*
RD1 U1-112 DrN1 0.119433
VD1 DrN1 DVn1 0
Hd1_2 DVn1 CCVSr1_2 CCVS VD2 0.0390498
Hd1_3 CCVSr1_2 CCVSr1_3 CCVS VD3 0.037191
Hd1_4 CCVSr1_3 CCVSr1_4 CCVS VD4 0.0364626
Hd1_5 CCVSr1_4 CCVSr1_5 CCVS VD5 0.0360892
Hd1_6 CCVSr1_5 CCVSr1_6 CCVS VD6 0.0359549
Hd1_7 CCVSr1_6 CCVSr1_7 CCVS VD7 0.0359489
Hd1_8 CCVSr1_7 CCVSr1_8 CCVS VD8 0.0358705
Hd1_9 CCVSr1_8 CCVSr1_9 CCVS VD9 0.0357969
Hd1_10 CCVSr1_9 CCVSr1_10 CCVS VD10 0.0356331
Hd1_11 CCVSr1_10 CCVSr1_11 CCVS VD11 0.0362686
Hd1_12 CCVSr1_11 CCVSr1_12 CCVS VD12 0.0356264
Hd1_13 CCVSr1_12 CCVSr1_13 CCVS VD13 0.0357708
Hd1_14 CCVSr1_13 CCVSr1_14 CCVS VD14 0.0358375
Hd1_15 CCVSr1_14 CCVSr1_15 CCVS VD15 0.0358785
Hd1_16 CCVSr1_15 CCVSr1_16 CCVS VD16 0.0358522
Hd1_17 CCVSr1_16 CCVSr1_17 CCVS VD17 0.0369935
Hd1_18 CCVSr1_17 CCVSr1_18 CCVS VD18 0.035952
Hd1_19 CCVSr1_18 CCVSr1_19 CCVS VD19 0.0379053
Hd1_20 CCVSr1_19 CCVSr1_20 CCVS VD20 0.0408726
Hd1_21 CCVSr1_20 CCVSr1_21 CCVS VD21 0.0362841
Mouse(mm): X: -23.750, Y: 10.885
Computing completed

Optimized Broadband Module

A typical workflow in Iteration Mode is the same as RLGC Module Pin-based Extraction. Additional steps in the Simulation Setup include:

1. Click:

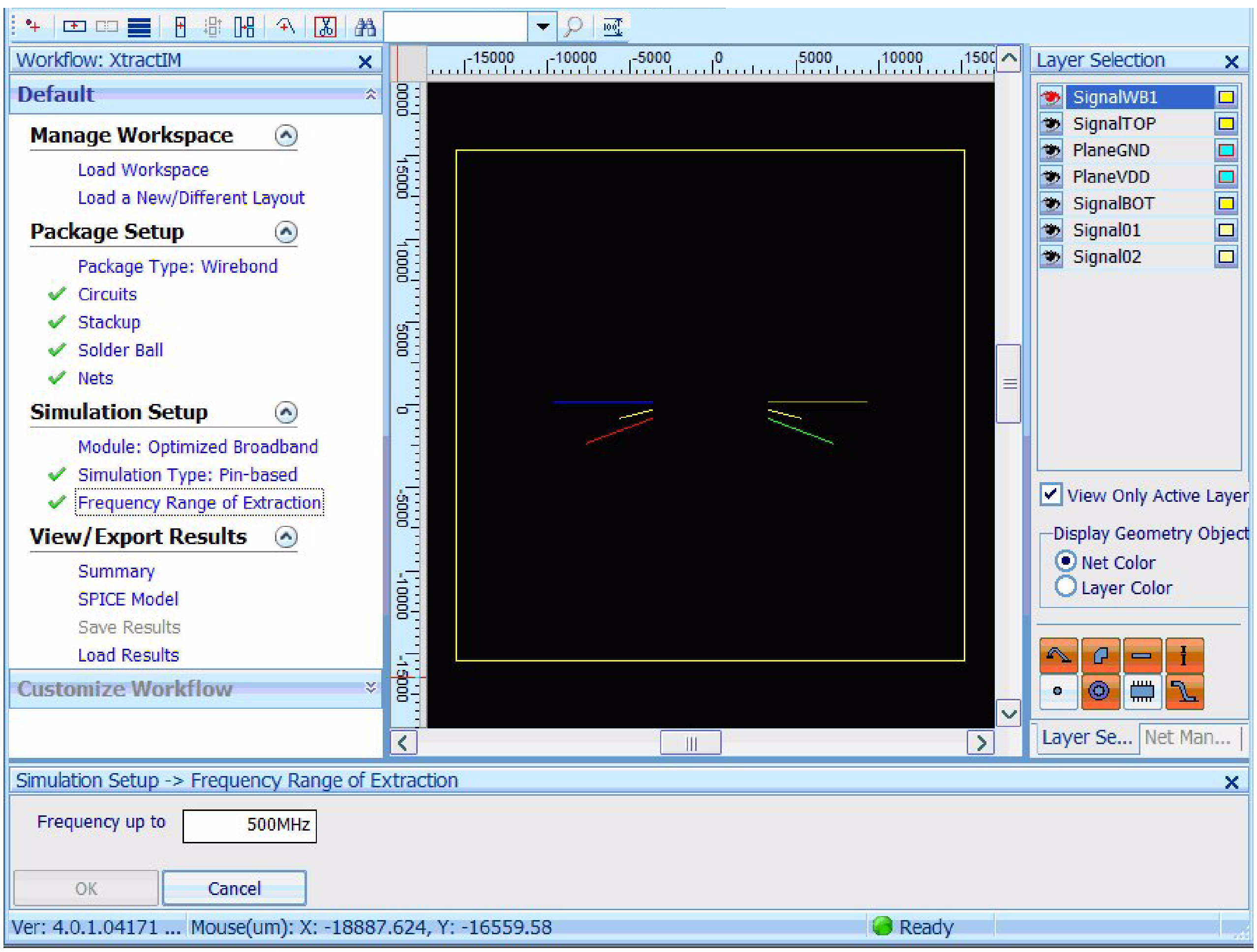
   Module > Optimized Broadband

2. Click:

   Workspace > Frequency Range of Extraction

   A new Edit pane opens at the bottom of the window.

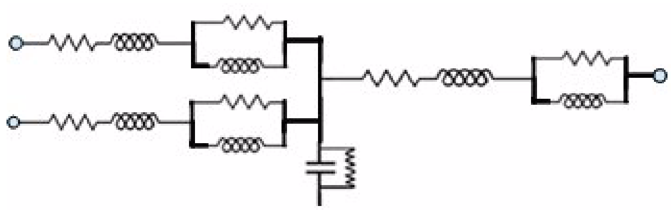
3. Input the highest frequency for simulation.



Circuit Topology

The circuit topology is a generalized T-model with series terms. No multi-sections circuit model is used as those in a net-based Broadband module.

Extraction frequency points for S-parameters are limited to a few points. Frequency band goes up to a few hundred MHz.



Chapter

7

IBIS/RLGC Module: Simulation of Leadframe Package

This chapter takes you through the steps to use the XtractIM tool in the simulation of Leadframe package.

PREPARE FOR SIMULATION

Collect this information before you begin the simulation.

- Make sure the Leadframe file is in DXF or SPD format
- Have the Stackup information ready
- Have the outer lead width, length, height and conductivity ready

If the leadframe file is in DXF format, make note of:

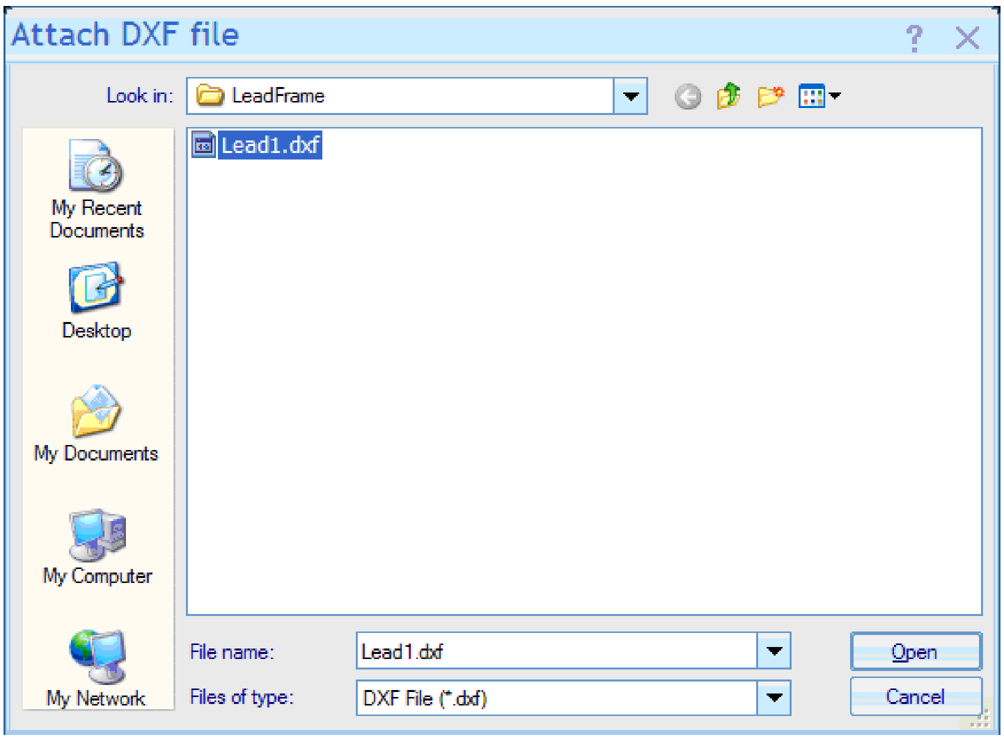
- Ball pad parameters
- Die pads
- Layers you wish to import
- Leadframe conductor Wirebonds

LESSON ONE : SIMULATION SETUP

1. Launch XtractIM.
2. Click **Open** to open an existing workspace.
3. Click:

   Load an existing layout
4. Click **OK**.

   The **Attach DXF File** window opens.
5. Select the desired file. The example shows only one file selected: **Lead1.dxf**.
6. Click **Open**.



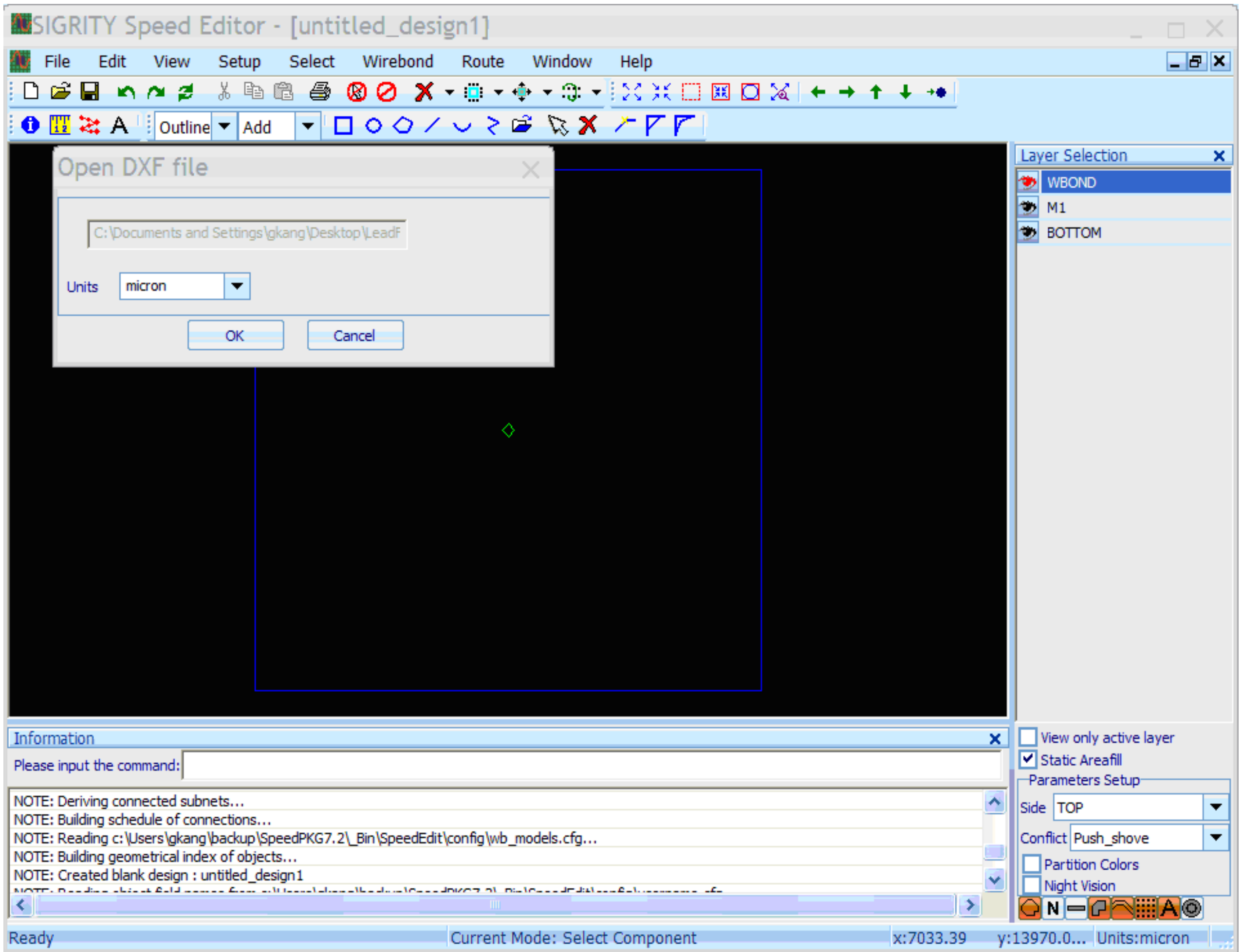
7. Launch XtractIM.
8. Click **New** to create a new workspace;

   or select:

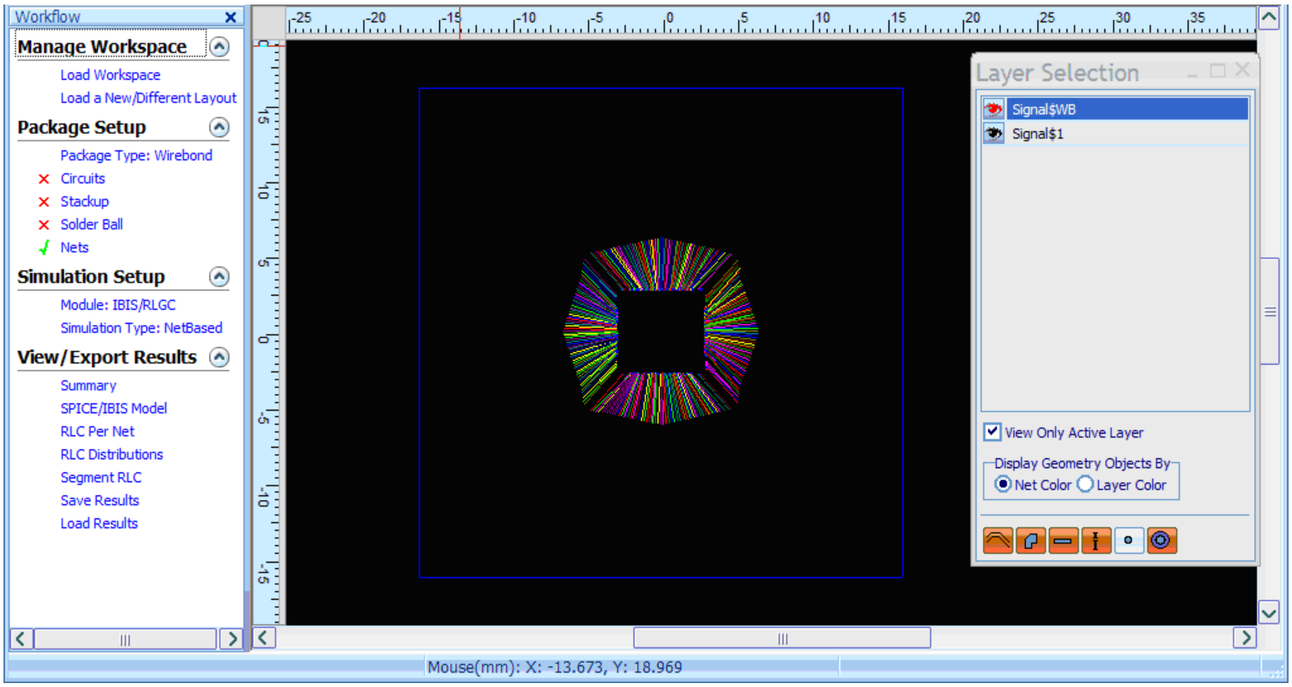
   Workspace -> New
9. To load the Package Structure, select:

   Load a New / Different Layout

10. Click **OK** to load a **DXF file**. The Sigrity Speed Editor is launched.



11. Translate a package file from DXF to SPD format; it is automatically loaded into XtractIM



Package Type

XtractIM classifies the leadframe family package into three kinds of packages:

- **No Outer Lead** — No board pin has outer lead. Typical package is QFN
- **Partial Outer Lead** — Some of the board pins have outer lead
- **With Outer Lead** — Every board pin has outer lead. Typical package is QFP

1. Select **Package Type**.

The default is Single-Die, Single-BGA, Wirebond Package.

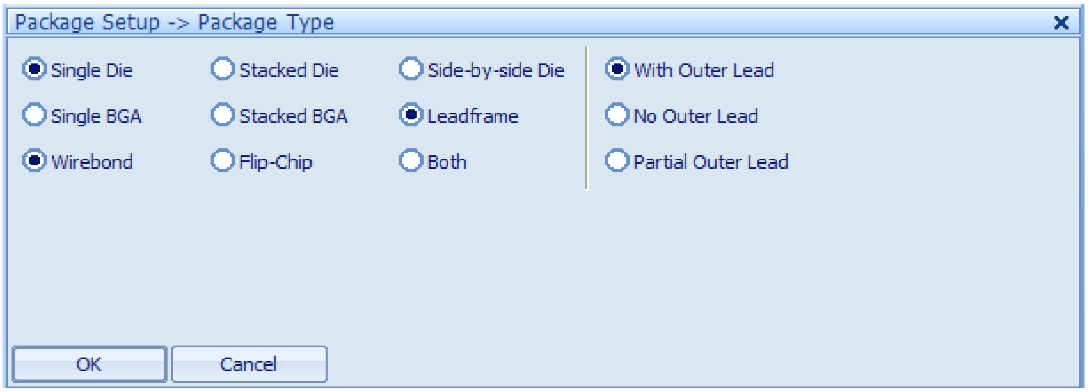
2. Click **Leadframe**.

3. Select one of:

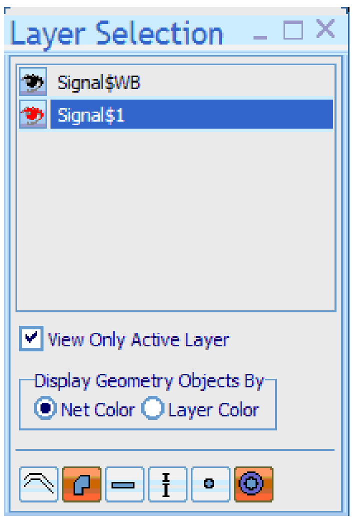
- No Outer Lead
- Partial Outer Lead
- With Outer Lead

4. Click **OK** to save your selection.

5. Click **Cancel** if you want to change your selection, start over, or cancel your session.



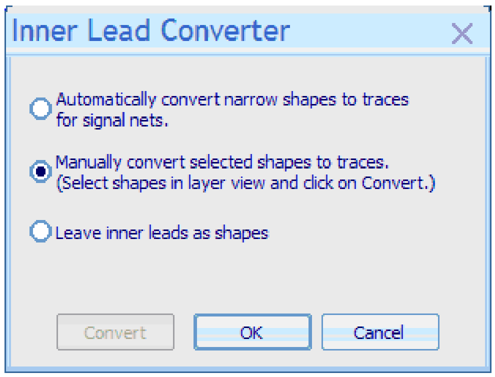
6. Click on the **Single$1** layer. The **Inner Lead Converter** locates **Single$1**.



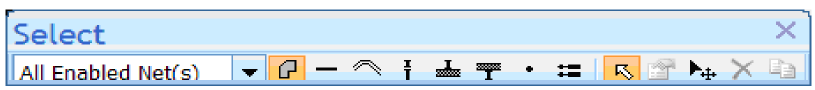
Shape to Trace Conversion

1. Click on **Inner Lead Converter** to convert narrow shapes into traces. The **Inner Lead Converter** window opens.
2. Choose:

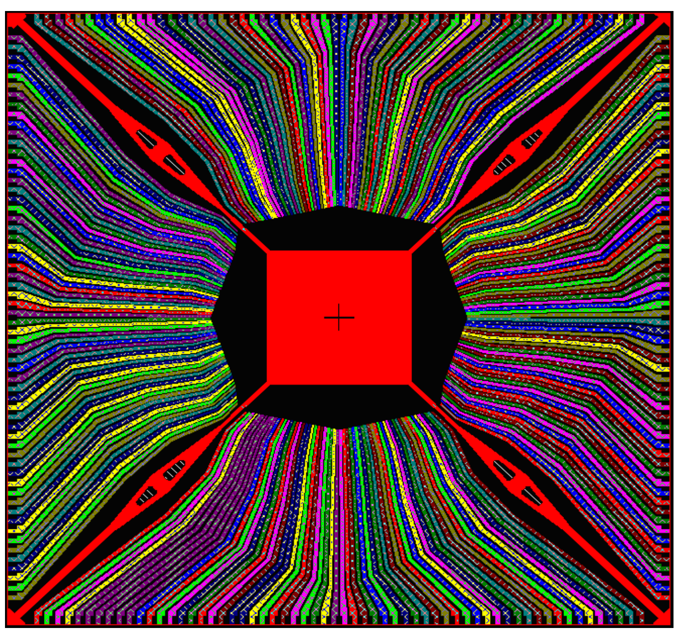
   Manually convert selected shapes to traces



3. Use the toolbar icon to select shapes in the Layer View.



4. Click **Convert**. All narrow shapes are converted into Traces.



NOTE!	A **No Outer Lead Package** generally does not need this converter.

Setup Circuits and Stackup

1. Click on **Circuits** in the Workflow pane to setup circuit data for the leadframe package. A new pane opens in the left side of the workspace.
2. Right-click on the desired circuits.
3. Select it as a **Die circuit**.
4. Click **Next**.
5. Right-click on another desired circuit.Select it as a **Board circuit**.
6. Click **Finish** to finish the circuits setup.
7. Click on **Stackup**.The Stackup window opens.
8. Right-click on the **Signal$** layer. A pop-up window opens.
9. Select:

   Insert Under

   A second pop-up menu opens.

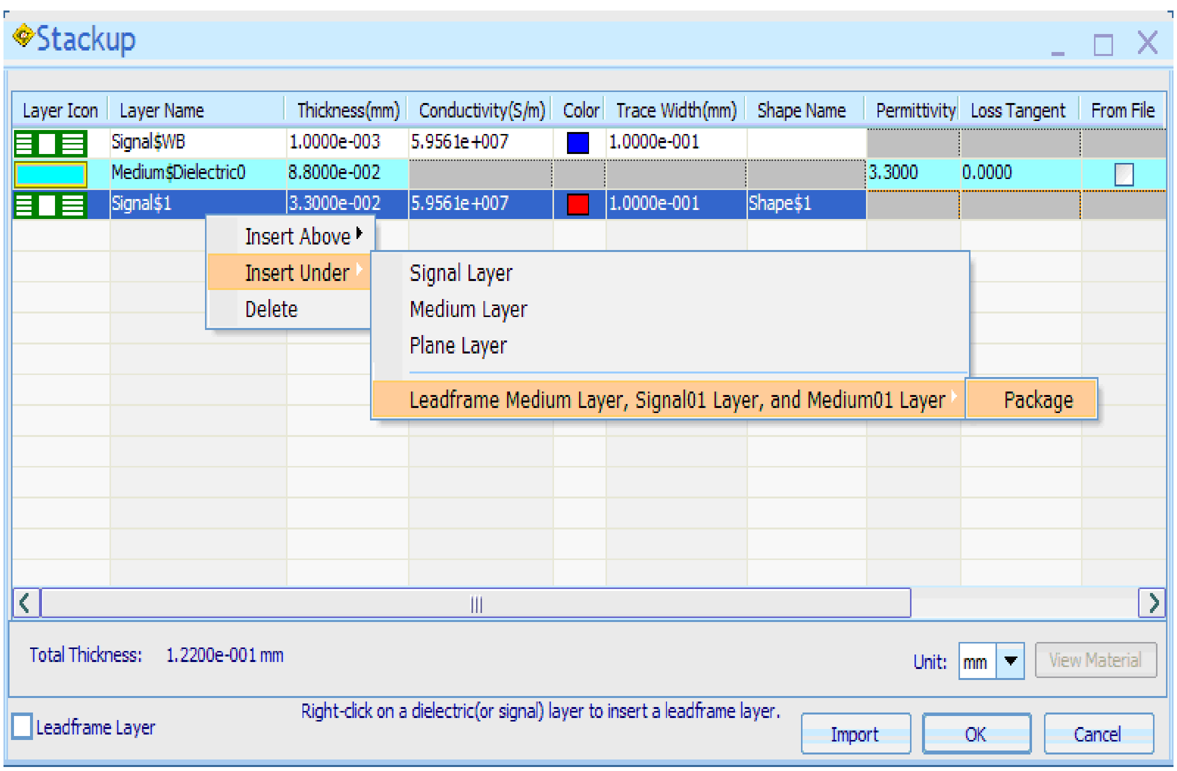
Outer Lead Setup

The following steps are performed for **Outer Lead** only.

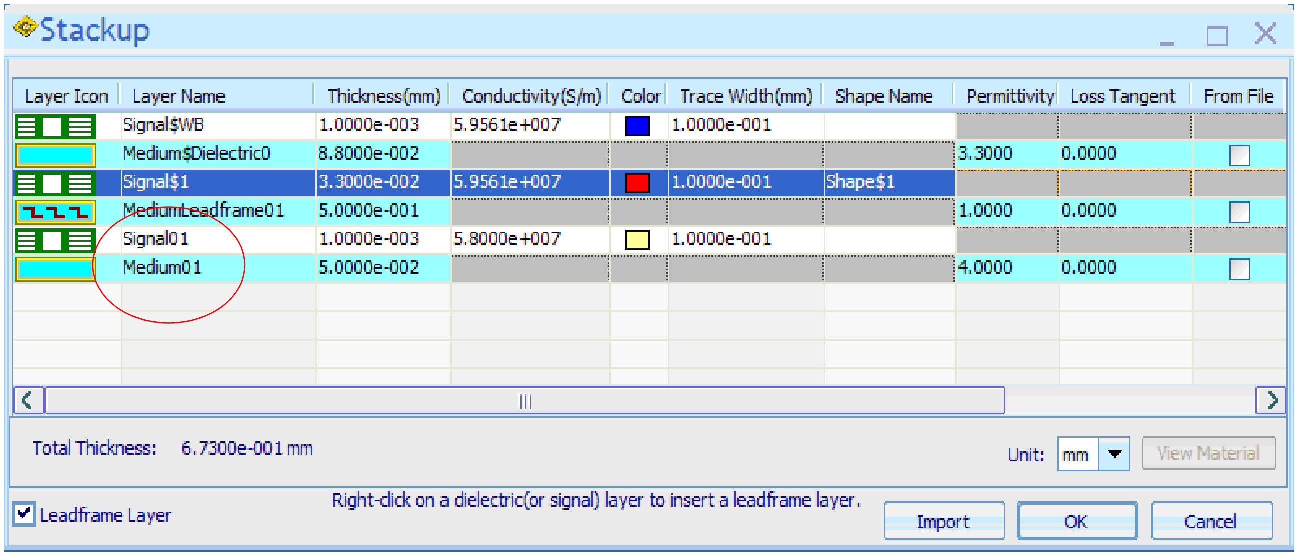
1. To insert a Leadframe layer, select

   Leadframe Medium Layer, Signal01 Layer, and Medium01 Layer

2. To insert an empty signal layer, select **Package**.



3. Insert a medium layer standing for a PCB medium layer.
4. All layers are inserted into **Signal$1**. The added signal layer is located at the end of the lead-frame.
5. Click **OK** to close the Stackup window.



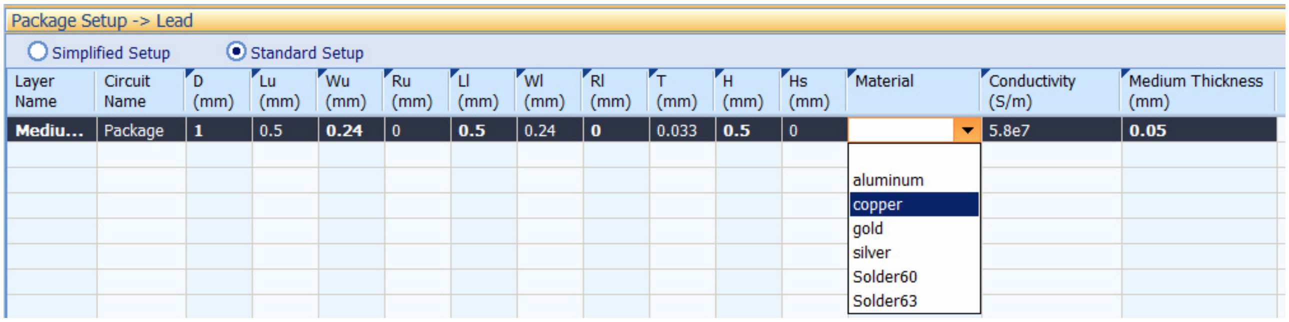
Leadframe Data for the Leadframe Package

1. Click on **Leadframe** in the Workflow pane to setup the data for the Leadframe package. A new pane opens up in the bottom portion of the windows.
2. Select:

   Package Setup -> Leadframe

3. Input the settings for Leadframe (Standard Setup).
   - **Conductivity** — Conductivity of the outer leads
   - **D** — Total horizontal length
   - **H** — Heights of the leads
   - **Hs** — Heights of the lower part
   - **LI** — Length of lower leads
   - **Lu** — Length of upper leads
   - **Medium** — PCB medium thickness
   - **RI** — Radius of lower arc
   - **Ru** — Radius of upper arc
   - **T** — Thickness of the leads
   - **WI** — Width of lower leads
   - **Wu** — Width of upper leads

Standard Setup Example



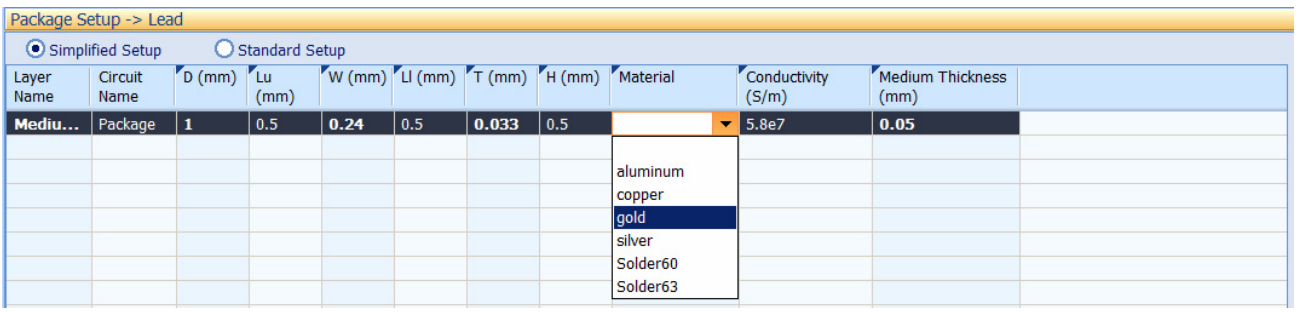
Package Setup -> Lead

Simplified Setup / Standard Setup

Layer Name	Circuit Name	D (mm)	Lu (mm)	Wu (mm)	Ru (mm)	Ll (mm)	Wl (mm)	Rl (mm)	T (mm)	H (mm)	Hs (mm)	Material	Conductivity (S/m)	Medium Thickness (mm)
Mediu...	Package	1	0.5	0.24	0	0.5	0.24	0	0.033	0.5	0		5.8e7	0.05

aluminum
copper
gold
silver
Solder60
Solder63

Simplified Setup Example



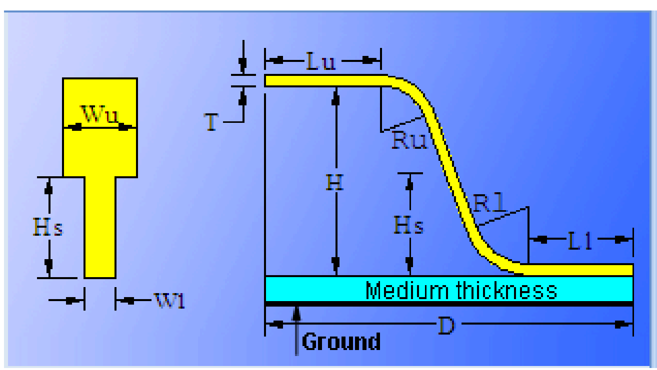
Package Setup -> Lead

Simplified Setup / Standard Setup

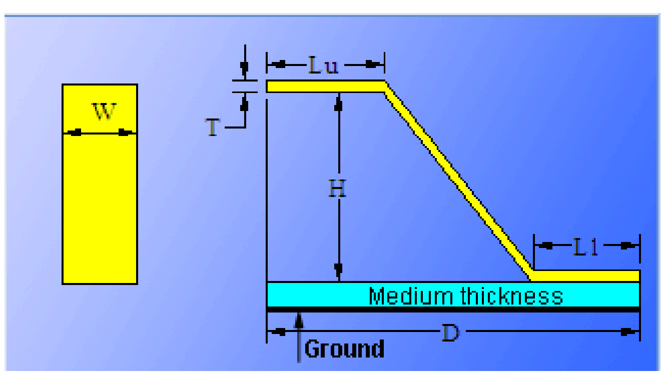
Layer Name	Circuit Name	D (mm)	Lu (mm)	W (mm)	Ll (mm)	T (mm)	H (mm)	Material	Conductivity (S/m)	Medium Thickness (mm)
Mediu...	Package	1	0.5	0.24	0.5	0.033	0.5		5.8e7	0.05

aluminum
copper
gold
silver
Solder60
Solder63

Standard Lead Plot Example



Simplified Lead Plot Example



4. Click **OK** to save your entries.
5. Click **Cancel** if you do not want to save your changes. You can start over.

Setup Nets

1. Click on **Nets** in the workflow window. The **Net Manager** window opens.
2. Choose any desired nets for RLC extraction.
3. Move the **signal net** into and out of PowerNets and Ground Nets, as needed.

   At least one Ground Net must be selected to act as a **reference ground net**.
4. Choose only the desired **Ground Net** as the reference ground net.

Extraction Frequency

1. To change the extraction frequency, select:

   Setup -> Frequency of Extraction

   A pop-up window opens.
2. Update the data in the pop-up window.
3. Use the Extraction Frequency window to change the default value. The default value is 30MHz.

Threshold for Exporting Mutual Terms

1. Use the XtractIM options to reduce the size of the output circuit during the export stage. XtractIM captures all the coupling during the extraction stage.
2. Open:

   Setup > Threshold for Exporting Coupling Terms

   A pop-up window opens.
3. Choose the number of strongest coupling neighbors to be kept in the circuit model.

   The default on number of strongest coupling neighbors is 10; which means only outputting the 10 strongest neighbors (including self).
4. Ignore mutual capacitance or inductance if the ratio of mutual terms over self term is less than a percentage.

   The default percentage threshold for ignoring mutual capacitance and inductance is 0.005.

   If the mutual capacitance or inductance is less than the 0.5% of the minimum of the two self-capacitances / inductances, then it does not output the mutual capacitance / inductance.

Lesson Two: Save Work Areas

Layout File

1. Click on the workspace.
2. Click on the **Open Layout File** icon (**green**) to open the project file.
3. Click on the **Save Layout File** icon (**green**) to save the project file.
4. To save the layout file under a different name, select:

   Save as

NOTE!	Saving the workspace automatically saves the .spd file. Saving the.spd file does not automatically saves the workspace.

Workspace

1. Click on the Workspace.The workspace opens.
2. Click on the **Save Workspace** icon (**yellow**) in the toolbar.
3. Use **Save as** to save the Workspace under a different name.

Lesson Three: Run the Simulation

1. Click on the **Play** button  at the top of the window to start the extraction (simulation).
2. XtractIM only extracts RLCG for a net which has at least one pin at the Die-side and at least one pin at the board side.
3. At the beginning of the simulation, if some nets have Die-Board mis-match, a pop-up window opens.
4. Select the next action.
   - **Continue** — Continue the simulation
   - **More Information** — Examine what nets are mis-matched
   - **Stop** — Cancel the simulation

Investigate Mis-matched Nets

The **More Information** window lists all the mis-matched nets.

1. Investigate the mis-matched nets to see whether it is a special design or a defective design.
2. Decide whether or not to proceed with the simulation.
3. Choose **Continue**, **Stop** or **More Information**.

   If 30 seconds pass and the user has not made a choice; then, by default, the simulation continues.

LESSON FOUR : OBSERVE AND SAVE SIMULATION RESULTS

XtractIM performs calculations for each net. The calculations include:

- Conductance
- Mutual capacitance with other nets
- Mutual loop inductance
- Resistance
- Self capacitance
- Self loop inductance

The inductance and capacitance are matrices.

In the Inductance Matrix, the Diagonal Element is the Self Inductance of each net.

The off-diagonal elements are mutual Inductance.

There are two concepts for capacitance matrix.

- Maxwell Capacitance Matrix
- SPICE Capacitance Matrix

Maxwell Capacitance Matrix — Each diagonal element is the loading capacitance

Example

Capacitance-to-ground when other nets are grounded. This represents the worst-case capacitive loading. Off-diagonal elements are mutual capacitance with negative values.

SPICE Capacitance Matrix — Each diagonal element is capacitance-to-ground. Off-diagonal elements are mutual capacitance with positive values.

View the relationship between Maxwell capacitance and SPICE capacitance matrix.

R, L and C

1. View **Resistance** for each net.
2. View **Self-inductance** for each net.
3. View **Self-capacitance** for each net.

Mutual Terms

1. View **Mutual Terms**.
2. View **Mutual Inductance**.
3. View **SPICE Mutual Capacitance**.

   Usually conductance is very low, so there is no view for conductance.

Summary of the Extracted Results

1. Click on **Summary**.
2. View the data for R, L and C.
   - Extraction Frequency
   - Full RLC Matrix
   - Maximum / Minimum R, L, C
   - Nets Extracted
   - Package Name
3. Reorder the list if you wish.
4. Click on **R/L/C** full matrix. The matrix is saved on hard disk in .csv format.

SPICE Model Files Saved

Upon completing the simulation, both model files are saved in the same directory as the .spd file.

The total number of elements (R, L, M, C and G where M is the mutual inductance) in the circuit is displayed at the bottom of the window.

The SPICE Model is saved as a SPICE sub-circuit with the extension .ckt.

The SPICE model is a T-circuit.

IBIS Model Saved

The IBIS Model is saved as an IBIS package model.

The saved model has the extension .pkg.

Pin Model: IBIS Format

An .ibs format file is saved.

The saved file includes each single net R, L and C.

All power nets and ground nets are lumped together.

Pin Model: Excel Format

1. Click on **Pin Model: Excel format** in the SPICE / IBIS Model window.
2. A **.csv** file is loaded. The **.csv** file includes information for each Signal Net.
   - Net Length
   - Net Name
   - Pin Name
   - Self-C
   - Self-L
   - Self-R
   - Time Delay

Self-C is the Maxwell Capacitance.

The information for Power Nets does not include Net Length.

DC Resistance

1. Click:

   DC Resistance

2. View the .csv file. DC Resistance is given for each of the Power, Ground and Signal Nets. A **.csv** file is saved on hard disk.

RLC Distributions

1. Click:

   RLC Distributions

2. View the full matrix value of R, L and C.

   RLC Distributions offers eight kinds of views.

3. View the RLC distributions.

Segment RLC

1. Click on **Segment RLC** in the workflow pane under:

   View / Export Results

   For each signal net, XtractIM outputs the segment RLC of each metal layer.

2. View the Segment RLC values. Note the three bars for Resistance, Inductance and Capacitance across the bottom of the screen.
3. Click on the **Resistance** bar. The Resistance values are displayed.
4. Click on the **Inductance** bar. The Inductance values are displayed.
5. Click on the **Capacitance** bar. The Capacitance values are displayed.

Save Simulation Results

1. In the Workflow pane click:

   Save Results

   The Save Extractor Result window appears.

2. Enter a file name.
3. Click on **Save**. The results are saved in a binary file. The file is named:

   result_spd_file_name.xim

4. Click on **Cancel** if you do not want to save the results in the file name you entered.

Simulation Output Files

The **result** and **result*.xim** files save all the output data including:

- Package Model Files.
- SPICE Circuit
- Summary

The SPICE file and two IBIS files are automatically saved by the tool.

The result file is created only when the user chooses to save all the output data.

View the output files on hard disk.

- One **IBIS Package Model File** — The *.pkg file includes both L and C coupling elements
- One **Pin Model in Excel Format** — The *.csv file includes each signal net length, self-R, self-L, self-C, and time delay. No coupling elements are included
- One **Pin Model in IBIS Format** — The*.ibs file includes each signal net self-R, self-L, and self-C. No coupling element is included
- One **Summary Content in Excel Format** — The *.csv file includes the RLC Full Matrix
- Two **SPICE Circuit Files** — The Pi-model is named *.ckt; the T-model is named *_t.ckt
- Three **Segment RLC in Excel Format** — This segment includes the RLC of each metal layer with *.csv files

Load in Saved Results

1. Select **Load Results** in the workflow pane to load the saved results. The main window shows the selected result.
2. View **present results** or the **loaded results**.
   - **Loaded Curves** — Shows the loaded results
   - **Present Curves** — Present extracted results
3. To unload a loaded result, click on the toolbar icon:

   Unload Extractor Result

LESSON FIVE: BATCH MODE SIMULATION

1. To run a simulation in batch mode, select:

   Start > Run

2. Change to the directory where the XtractIM.exe file is located.
3. Upon completing the simulation, all output files are automatically saved in the same directory as the *.spd file.
   - *.ckt files
   - *.ibs files
   - *.pkg files
   - .csv files

Chapter

8

Electrical Performance Assessment

This chapter takes you through the steps to use the XtractIM tool to analyze the electrical performance of the Power / Ground / Signal distribution system. XtractIM checks the Power / Ground distribution system by checking:

- Power-ground Net Pair: Inductance, Capacitance and Broadband Impedance
- Per pin Resistance and Inductance
- Current Density: Via Current, Plane Current Density, Voltage Distribution (PowerDC license required)

XtractIM checks the Signal Distribution System by checking:

- Wirebond / Trace Layout: Impedance and Coupling
- Net Couplings: Mutual LC and Near-ended Crosstalk
- Insertion and Return Loss

XtractIM can also create Html report including all features in electrical performance assessment:

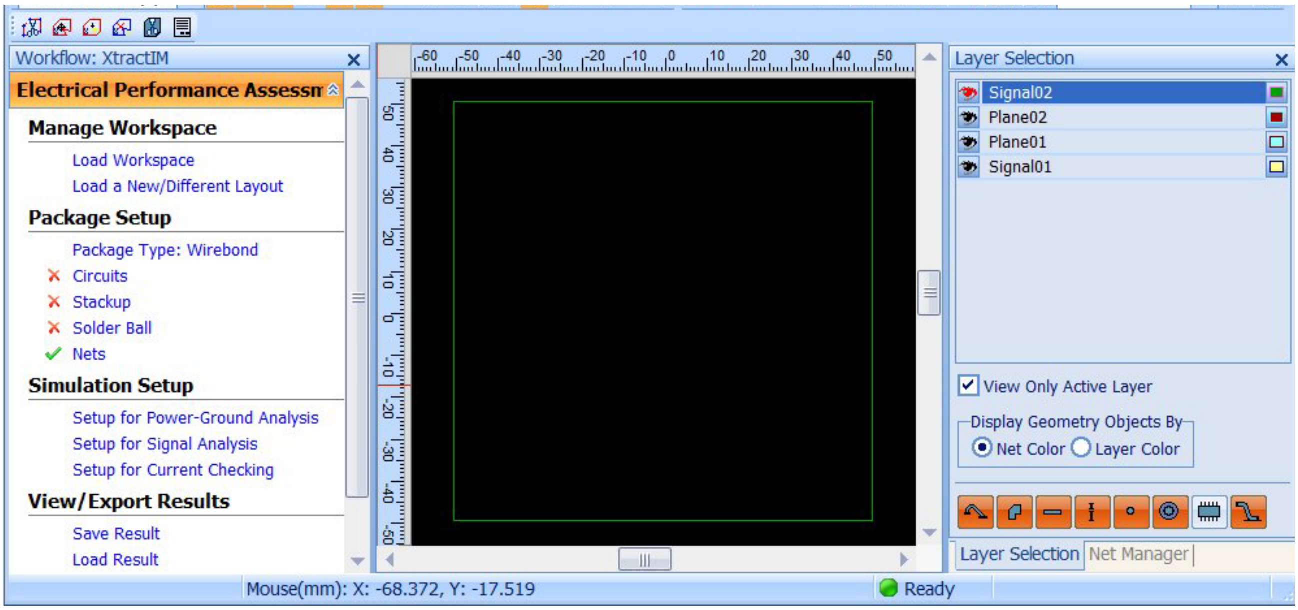
- Html report can be manually created after the simulation is completed
- Html report can be automatically created upon the simulation's completion

LESSON ONE: SIMULATION SETUP

1. Launch **XtractIM**.
2. Select:

   Mode > Elec. Perf. Assessment (EPA)

   The **Electrical Performance Assessment Workflow** appears.



Setup Steps

Follow these steps to perform a **Type Workflow** in **Interactive Mode**.

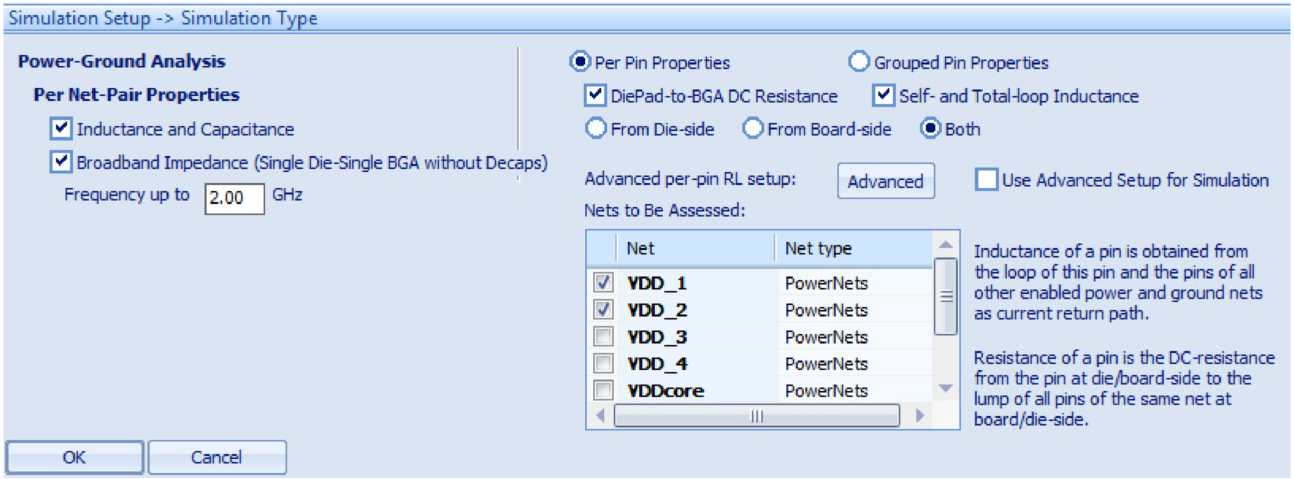
1. Load an existing file (.xml file), if any.
2. Open a layout file (.spd file).
3. Select a package type.
4. Setup the circuits.
5. Select or deselect Die-circuit and Board circuit.
6. Setup the Stackup.
7. Set parameters for the Bump or Solderball Medium layer.
8. Set the Bump data if it is a Flip-chip package.
9. Set the Solderball data.
10. Select the nets for simulation.
11. Simulation Setup.
12. Select automatically generate and save report.
13. Save the Workspace and Layout file.

Except for Step 11 and 12, all steps are exactly the same as the steps described in ***IBIS / RLGC Module: Net-Based Simulation Single-Die Single-BGA Packages***.

Setup for Power/Ground Analysis

1. Select:

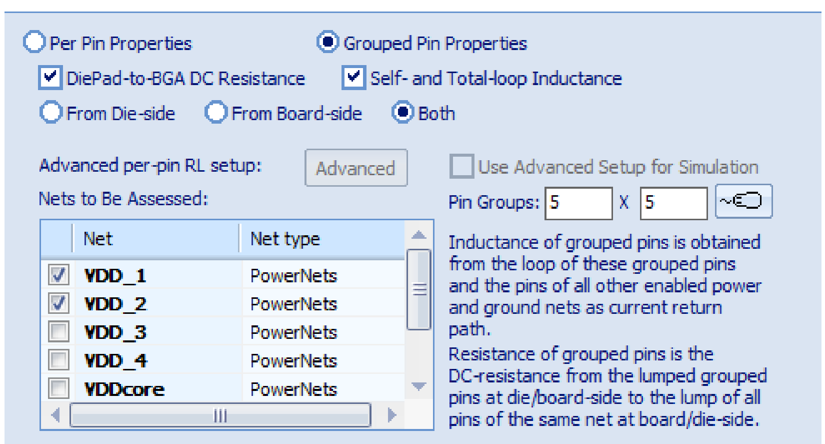
   Simulation Setup > Setup for Power / Ground Analysis



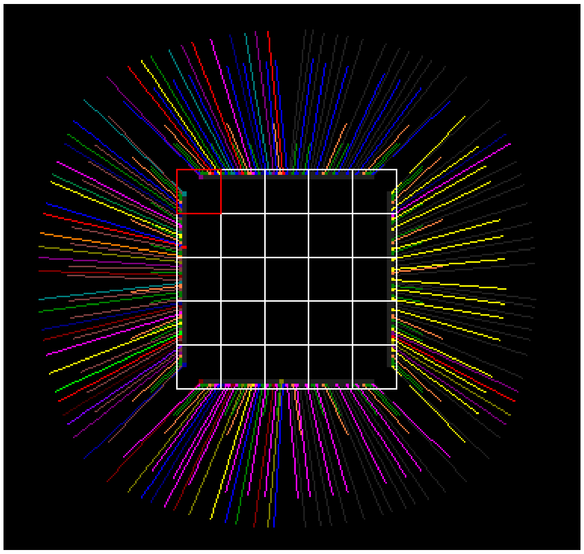
2. Select any or all of the following:
   - (Power to Ground) **Inductances and Capacitances**
   - **Broadband Impedance (Single Die-Single BGA without Decaps)** — Edit the highest frequency
   - **Bump-to-BGA DC Resistance** — Select Nets to be assessed
   - **Self- and Total-loop Inductance** — Select Nets to be assessed
3. Click **OK** to accept the selections.

Grouped Pin Property

If **Grouped Pin Property** is selected, user can choose n* n groups.



For example, enter 5 * 5 and it will give 25 cell as the following figure shows.

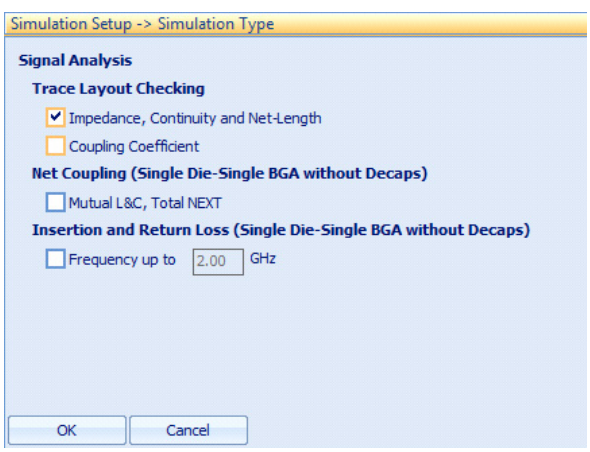


In each of the cells, instead of per pin assessment, all pins of the under assessment net will be lumped and assessed as one. The return paths are kept the same as per-pin properties assessment.

Setup for Signal Analysis

1. Select

   Simulation Setup > Setup for Signal Analysis



2. Select any or all of the following:

   Trace Layout Checking

   Impedance, Continuity and Net-Length: Before running coupling and insertion and return loss analysis, it is helpful to run this check first to generate a length report for filtering out the worse nets for insertion and returen loss analysis, which is usually completed in seconds.

Signal Net Name	Net Length (mm)
FCHIP_A1	2785.115
FCHIP_A2	3663.313
FCHIP_A3	4550.800
FCHIP_A6	3442.505
FCHIP_A7	2442.132
FCHIP_A12	2484.810
FCHIP_A13	3407.297
FCHIP_A16	4623.287
FCHIP_A17	3678.313
FCHIP_A18	2785.115
FCHIP_B3	2865.140
FCHIP_B4	3758.338
FCHIP_B5	4659.820
FCHIP_B6	3613.363
FCHIP_B13	3671.353
FCHIP_B14	4651.536
FCHIP_B15	3758.338

Coupling Coefficient

Net Couplings (Single Die-Single BGA without Decaps)

Mutual L&C, Total NEXT

Insertion and Return Loss (Single Die-Single BGA without Decaps)

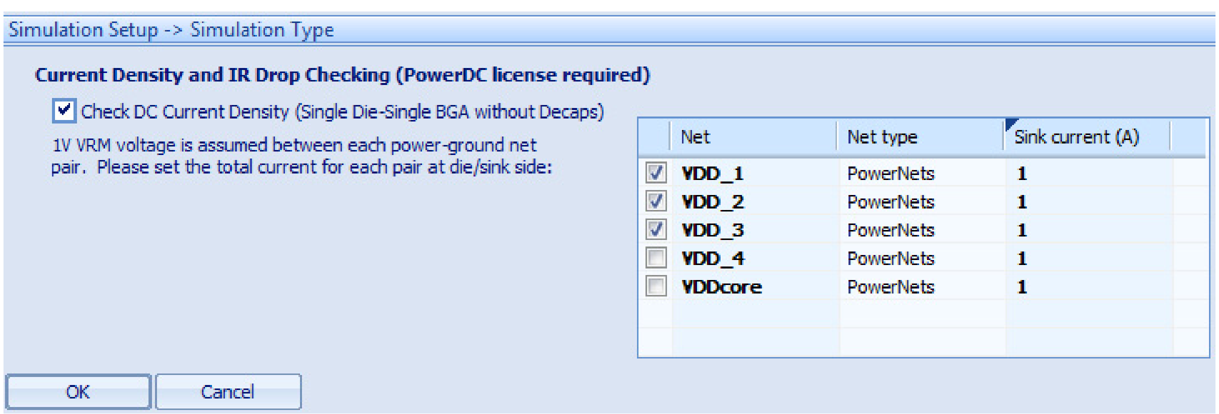
Frequency up to ____ GHz

Insert the highest frequency in the GHz box.

3. Click **OK** to accept the selections.

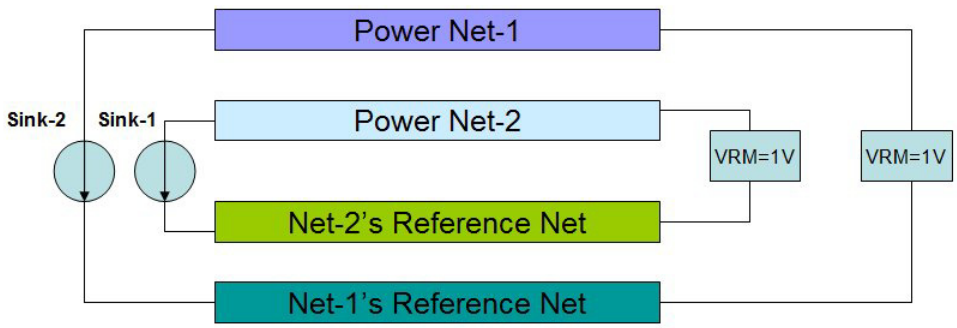
4. Select

Simulation Setup > Setup for Current Checking



- **Check DC Current Density (Single Die-Single BGA without Decaps)** — Every VRM voltage is set as 1V internally

5. Set the **Sink Current**. Sink current is set for each power net and its reference ground net. The simulation will be based on equal voltage at the Sink Side.

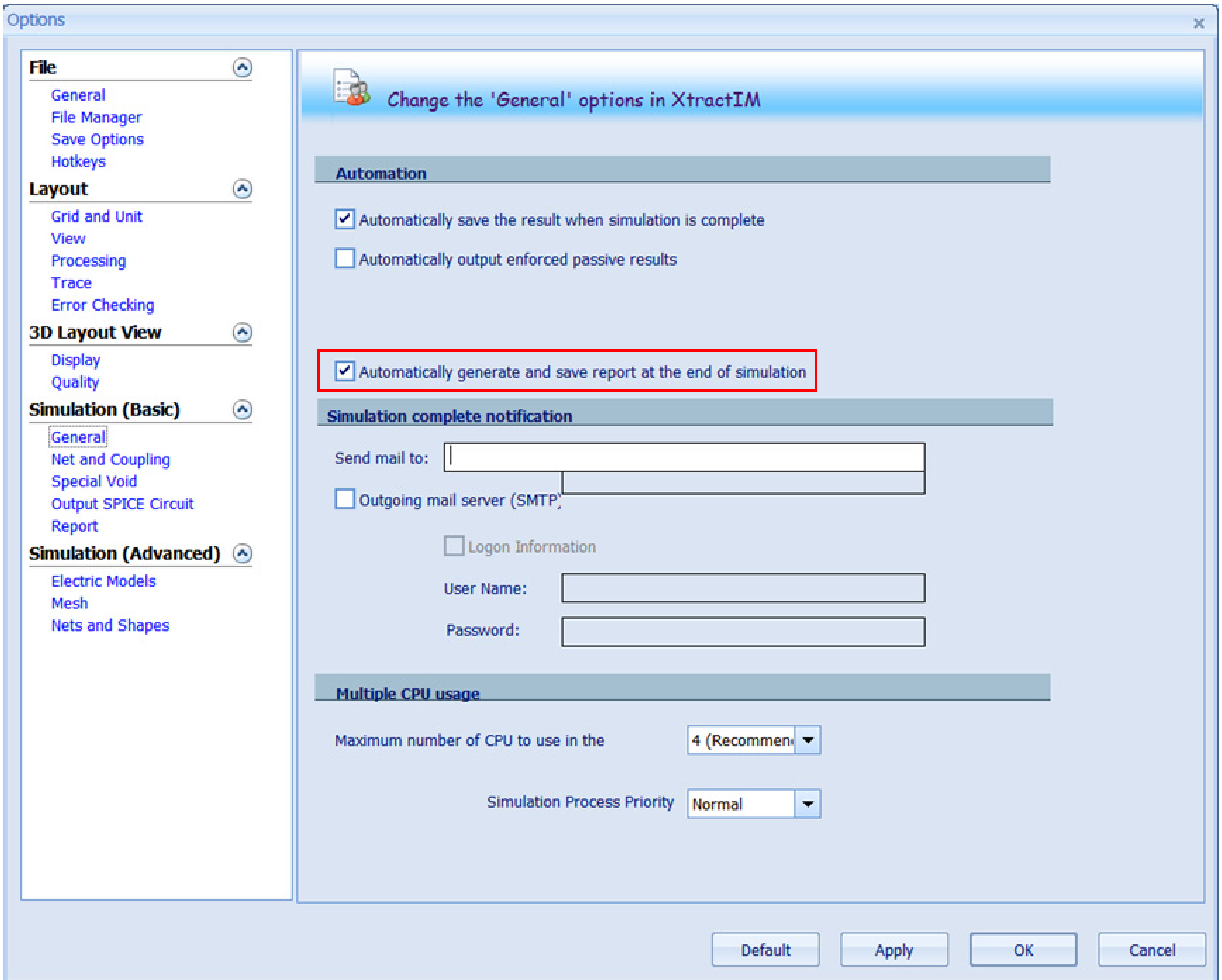


6. Click **OK** to accept the selections.

Select Automatically Generate and Save Report

1. To automatically generate and save report upon simulation's completion, select

Tools > Options > Edit Options... > Simulation (Basic) > General



2. Select the **Automatically generate and save report at the end of simulation** option.
3. Click **OK**.

Save Workspace

1. To save the workspace, click:

   Workspace > Save

   or

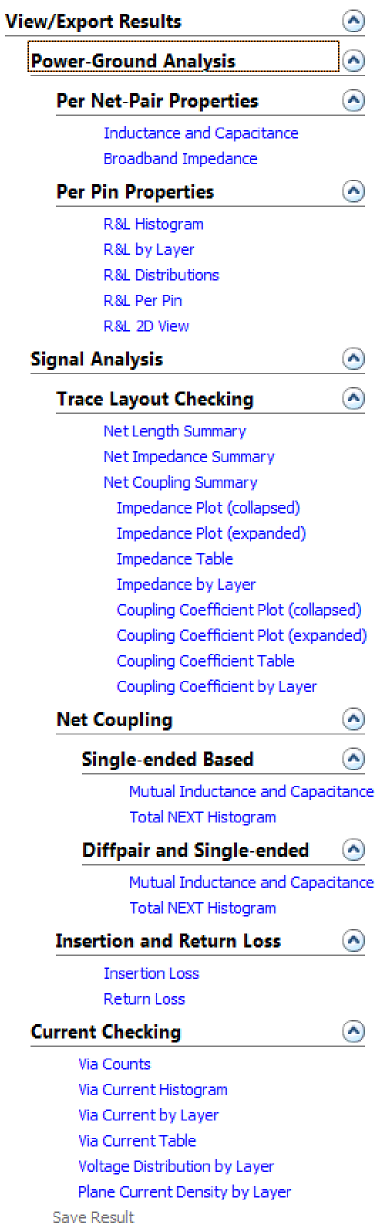
2. Click  to save both workspace and SPD file.

Lesson Two: Run and Simulation

Click on the **Play** button  at the top of window to start the extraction.

Lesson Three: Observe and Save Simulation Results

The results are displayed after the simulation is finished.



Power-Ground Analysis

Per Net-Pair Property

The Workspace lists the options to display all results. Click on the desired option.

Per Net-Pair Properties

Inductance and Capacitance
Broadband Impedance

1. Click:

   Inductance and Capacitance

   A table is displayed.

Power Net	Ref Ground Net	L(nH)	C(pF)
VDD_1	VSS	0.454952	43.3595
VDD_2	VSS	0.458492	43.3065
VDD_3	VSS	0.451241	43.4001
VDD_4	VSS	0.455836	43.2714

Ground Net	L(nH)
VSS	0.454952
VSScore	0.590788

- **Power Net** — Power Net Names
- **Ref Ground Net** — Reference Ground Net Name of smallest loop Inductance.
- **L(nH)** — Loop Inductance referring to the Ref Ground Net.
- **C(pF)** — Capacitance referring to the Ref Ground Net.
- **Ground Net** — Ground Net Name. Displays when Net Name is clicked.
- **L(nH)** — Inductance when the Power Net uses each Ground Net as the current return path. Displays when Net Name is clicked.

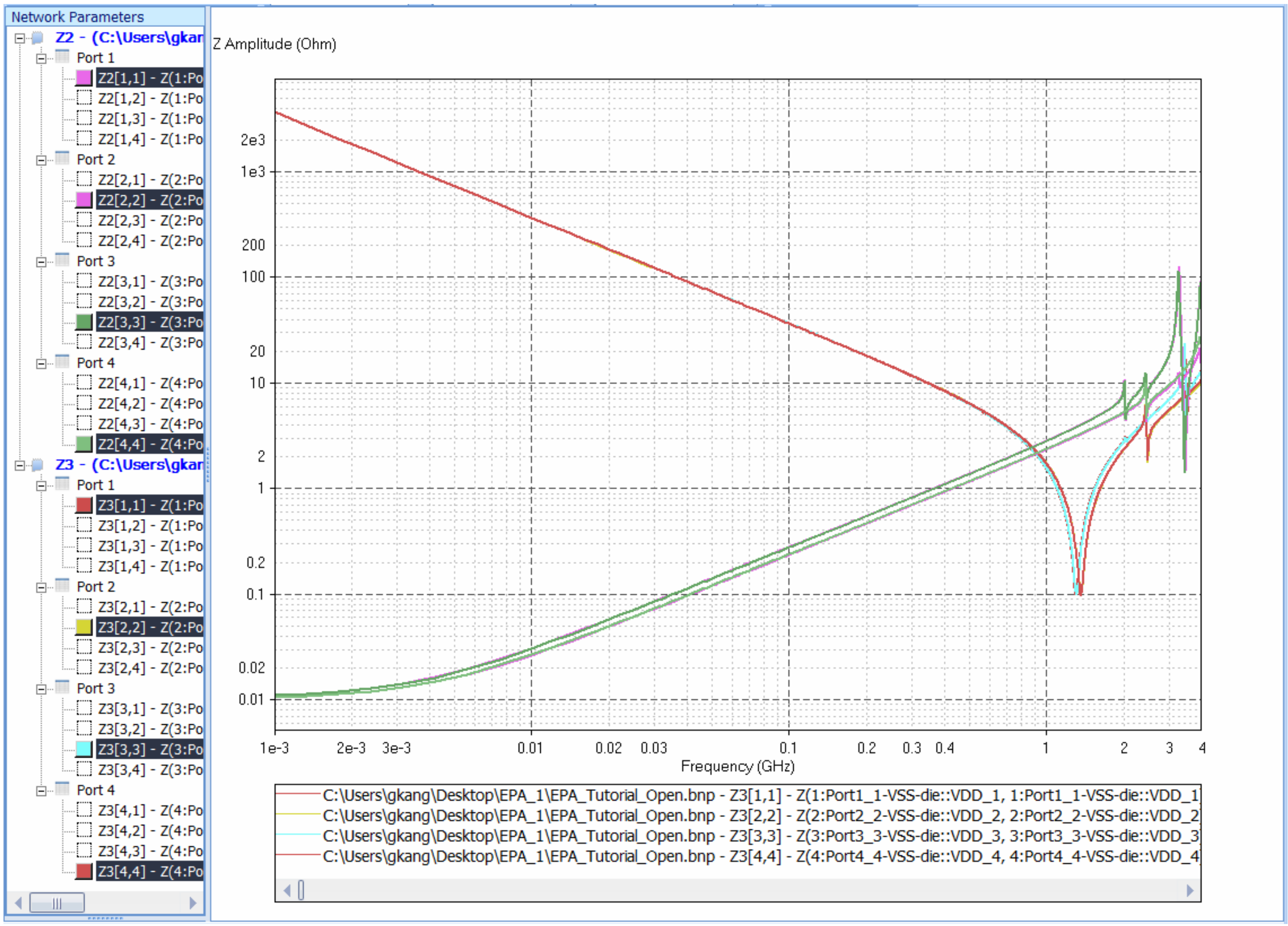
2. Click:

   Broadband Impedance

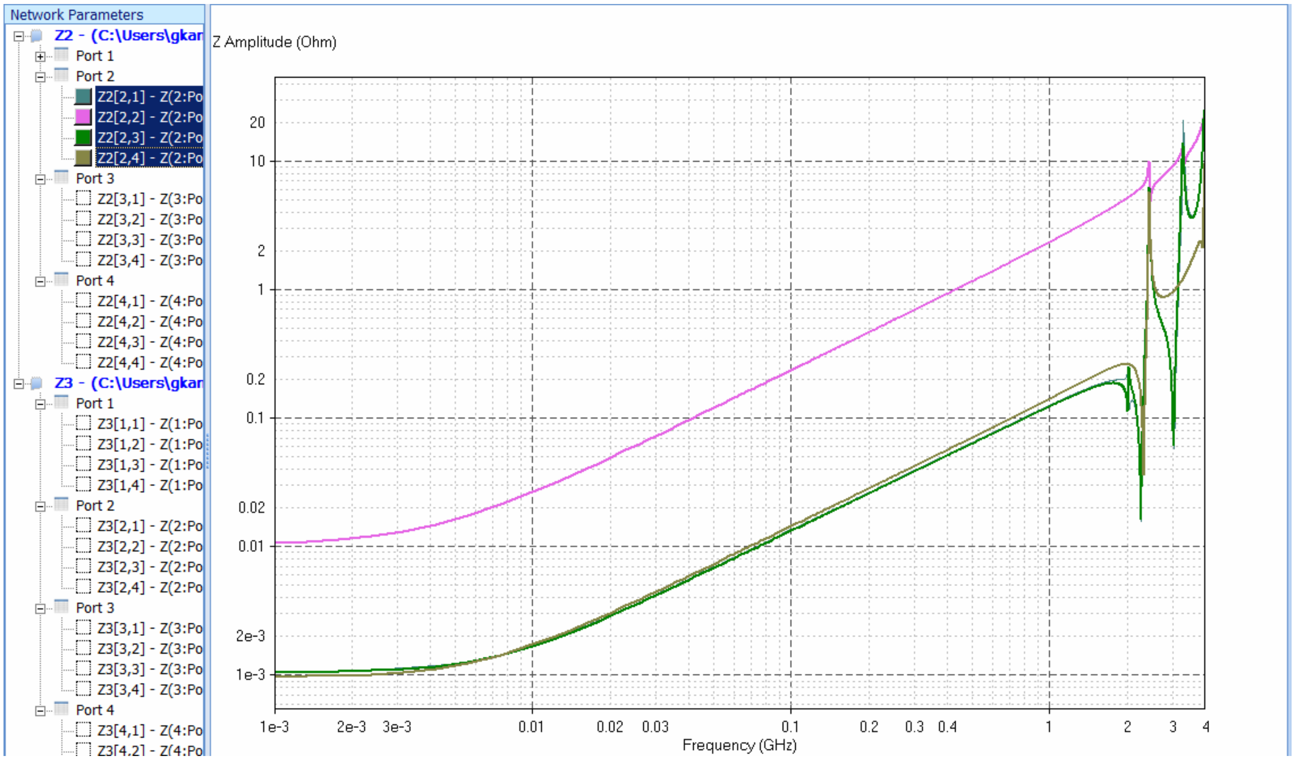
   Short and **Open Impedance** (Z-parameter) vs. **Frequency** (1MHz-Fmax) are shown in the same plot. The **Observation Ports** are at Die Side. Each Port is defined as a Power Net refers to its Reference Ground Net.

Definition of Short and Open

- **Short** — Short each Power Net and its Reference Net at the Board Side.
- **Open** — Keep its Power Net and its reference Net open at the Board Side.



Enable all curves of one port, both the self-impedance and transfer impedance are shown.



Per Pin Property

The Workspace lists the options to display all results. Click on the desired option.

Per Pin Properties

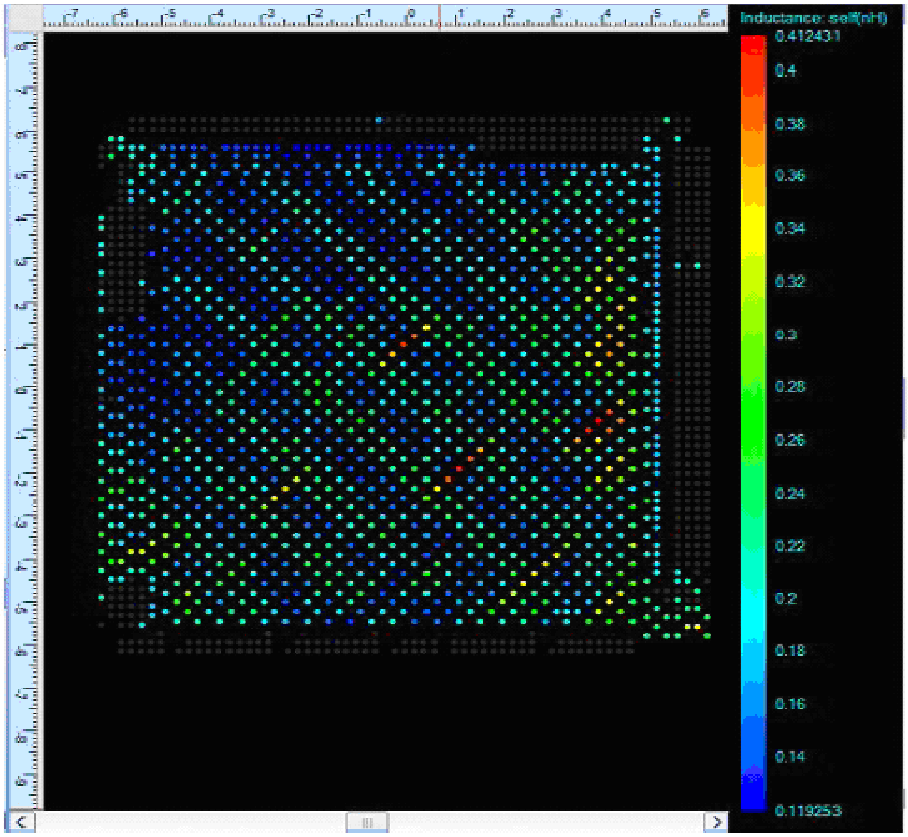
- R&L Histogram
- R&L by Layer
- R&L Distributions
- R&L Per Pin
- R&L 2D View

R&L Histogram

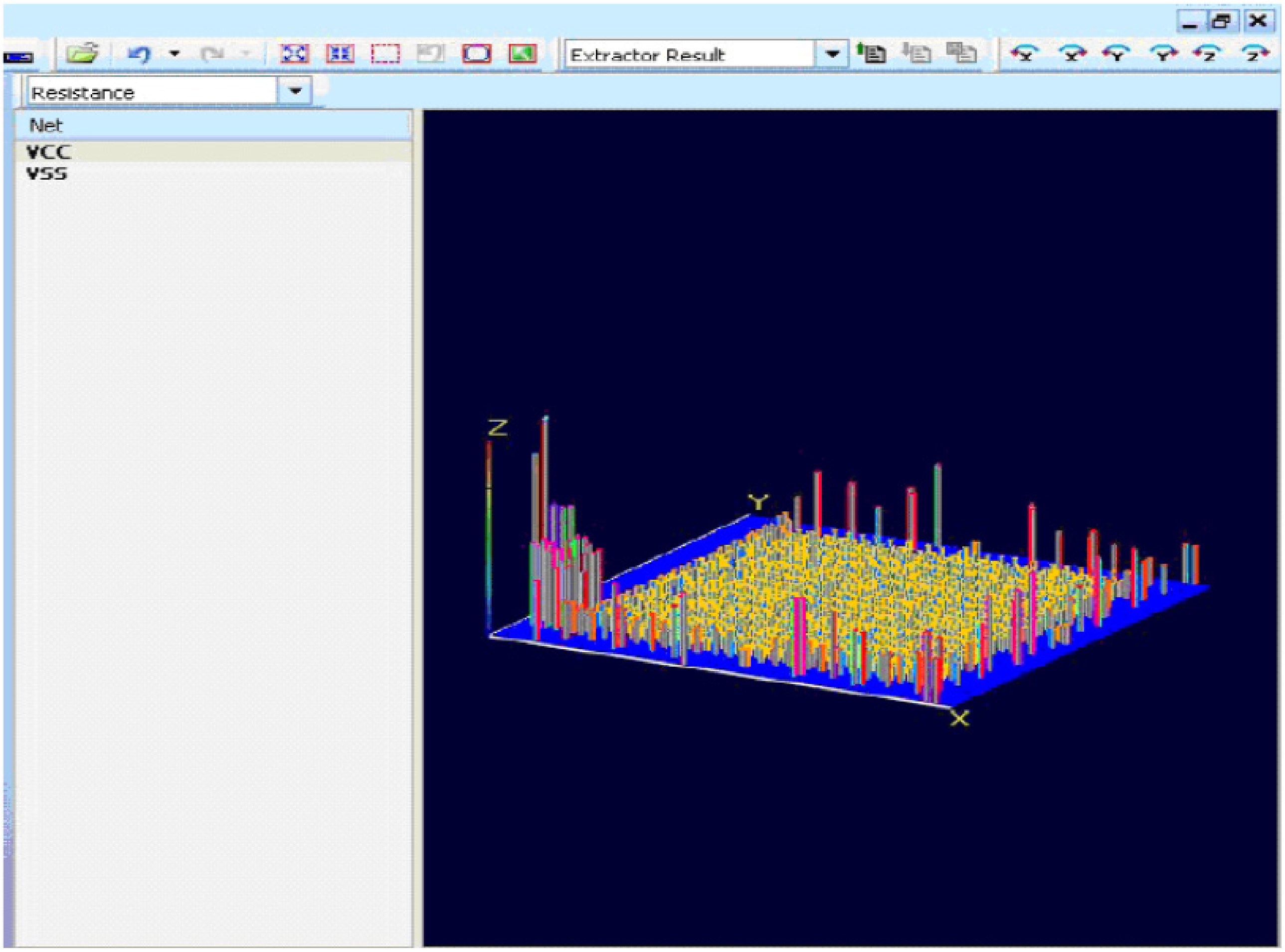
All pin RL values are plotted from the smallest (left) to the larget.

R&L by Layer

Inductance or resistance values are super-displayed in the layout.



R&L Distributions



- Use the **Rotation** icon to choose the preferred view.
- To save a file to any preferred name, select the name and click **Save**.

R&L Per Pin

Net
FCHIP
VDD
VDD_15
VSS
BGA
VDD
VDD_15
VSS

Pin NodeName	Self L(nH)	Total L(nH)	R(mOhm)	Induced Voltage(V)	Current(A)
Node03060!!A2::VDD	1.872	2.332	6.151	1.466	1.000
Node03069!!A11::VDD	1.845	2.315	6.149	1.454	1.000
Node03097!!D3::VDD	1.099	1.558	3.289	0.979	1.000
Node03104!!D10::VDD	1.071	1.534	2.966	0.964	1.000
Node03112!!E6::VDD	0.694	0.975	2.508	0.613	1.000
Node03114!!E8::VDD	0.817	1.301	14.615	0.818	1.000
Node03124!!F6::VDD	0.700	1.024	14.178	0.643	1.000
Node03126!!F8::VDD	0.784	1.227	14.229	0.771	1.000
Node03136!!G6::VDD	0.761	1.212	14.036	0.762	1.000
Node03138!!G8::VDD	0.728	1.078	14.593	0.677	1.000
Node03148!!H6::VDD	0.813	1.238	14.529	0.778	1.000
Node03150!!H8::VDD	0.792	1.176	14.647	0.739	1.000
Node03157!!J3::VDD	1.053	2.196	3.233	1.380	1.000
Node03164!!J10::VDD	0.961	1.968	2.931	1.237	1.000
Node03170!!K4::VDD	1.288	2.423	34.876	1.522	1.000
Node03175!!K9::VDD	1.190	2.190	34.574	1.376	1.000
Node03192!!M2::VDD	1.993	2.362	6.120	1.484	1.000
Node03201!!M11::VDD	1.634	2.040	6.132	1.282	1.000

- Column 1 - Net names
- Column 2 - Pin node name of the selected net
- Columns 3 and 4 - DC Resistance and Loop Inductance of each pin
- Columns 5 and 6 - Induced Voltage and Current of each pin

  These two columns provide a current-weighted per-pin power/ground RL assessment before running a system-level PI/SI analysis if you have a current vector.

  - Induced Voltage is calculated by dv=L*(di/dt)
  - The induced Current is calculated by $[L\_matrix]_{N*N}*[Current]$

    N is pin number for a net. By default, all the pins have the same current value **1A**.

1. Click the **E** button in the Current column of a pin, and input the pin Current value.

   The Induced Voltage result will be updated instantly.

Induced Voltage(V)	Current(A)
2.054	1.500 E
1.527	1.000
0.995	1.000
0.972	1.000
0.620	1.000
0.834	1.000
0.645	1.000
0.777	1.000
0.763	1.000
0.679	1.000
0.779	1.000
0.740	1.000
1.383	1.000
1.238	1.000
1.526	1.000
1.378	1.000
1.485	1.000
1.282	1.000

2. Click:

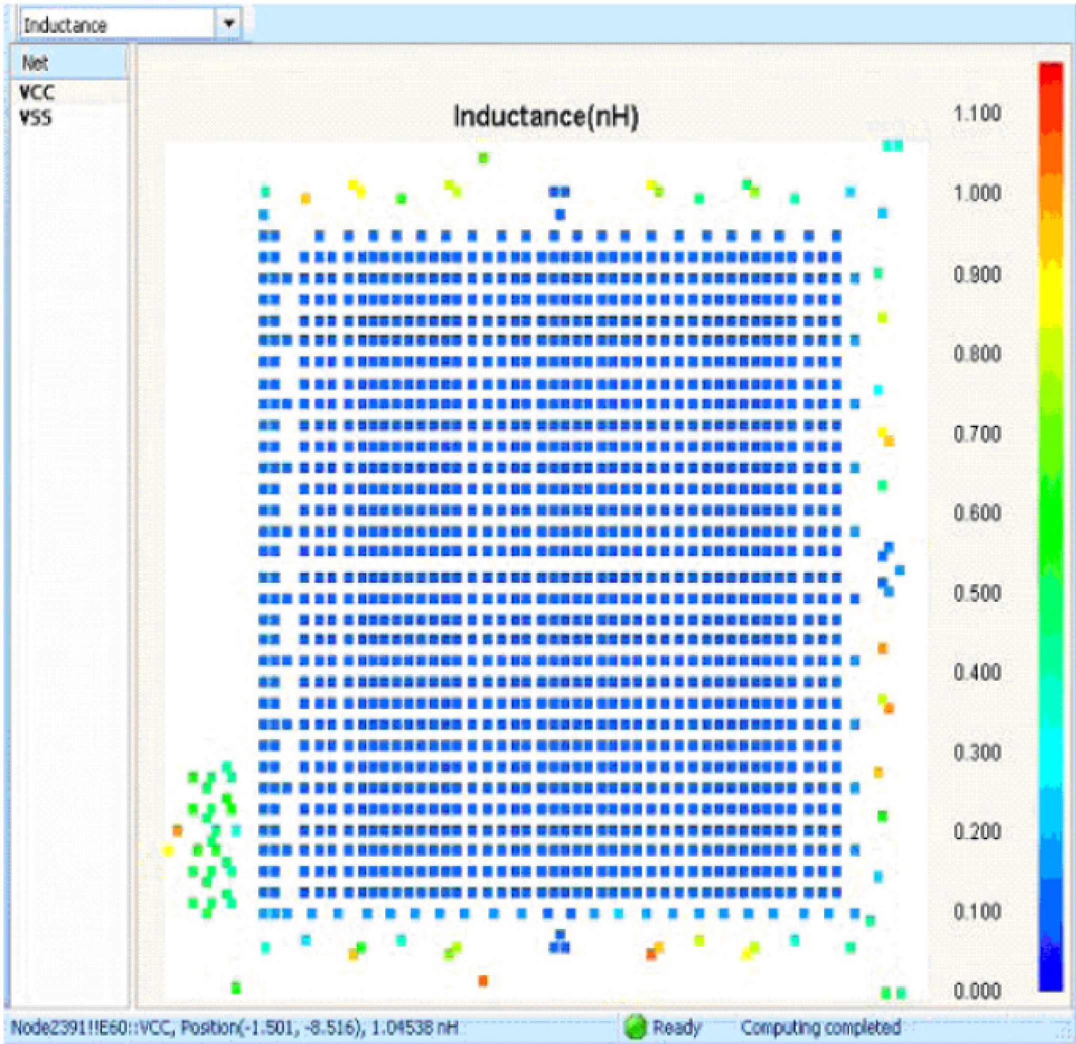
   Pin Node Name

3. Sort the list in ascending or descending order.
4. Click:

   R(mOhm) or L(nH)

5. Sort the items in ascending or descending order. The results are displayed in a table.

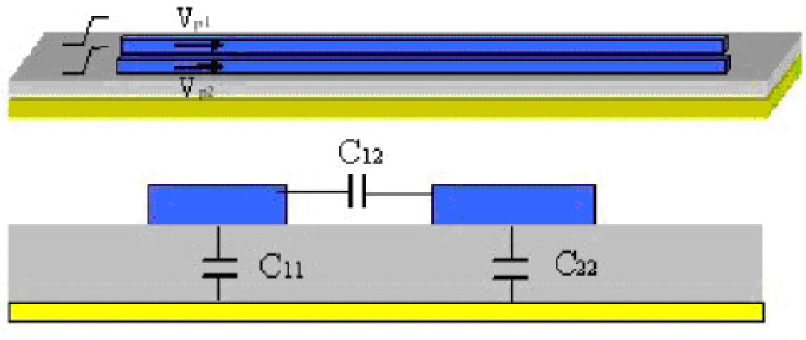
R&L 2D View



1. Select from the drop-down menu to see **Resistance** or **Inductance**.
2. Move the cursor across the Layout window.
3. Pause the cursor on a point of interest. The display shows:
   - Node Name
   - X-Y coordinates
   - Actual number

Signal Distribution System Checking

Trace coupling is defined with Near-ended Crosstalk as a victim.



Aggressor: Z_{10}, L_{11}, C_{11}

Victim: Z_{20}, L_{22}, C_{22}

$$K = \frac{V_p}{4}\left(C_{12}Z_{20} + \frac{L_{12}}{Z_{10}}\right)$$

$$\text{Where, } \frac{2}{V_p} = \frac{1}{V_{p1}} + \frac{1}{V_{p2}}$$

$$V_{p1} = \frac{1}{\sqrt{L_{11}C_{11}}} \quad V_{p2} = \frac{1}{\sqrt{L_{22}C_{22}}}$$

$$Z_{10} = \sqrt{\frac{L_{11}}{C_{11}}} \quad Z_{20} = \sqrt{\frac{L_{22}}{C_{22}}}$$

Definitions

K	The near-end coupling coefficient from aggressor Net on victim Net.
C_{11} / C_{22}	The loading capacitance per unit length of Signal Net to its Reference Net.
L_{11} / L_{22}	The self-inductance per unit length of Signal Net.
C_{12}	The mutual capacitance per unit length between aggressor and victim Net.
L_{12}	The mutual inductance per unit length between aggressor and victim Net.
V_{p1} / V_{p2}	The signal velocity propagates on Signal Net.
Z_{10} / Z_{20}	The characteristic impedance of Signal Traces.

Trace / Wirebond Layout Checking

The Workspace lists all results to be displayed. The first three summaries are for all enabled **Signal**

Nets in the package.

The rest of the summaries are for a specific path from one component to another component. For the Single-die Single-BGA package, the path is from Die to Board.

Signal Analysis

Trace Layout Checking

Net Length Summary
Net Impedance Summary
Net Coupling Summary
Impedance Plot (collapsed)
Impedance Plot (expanded)
Impedance Table
Impedance by Layer
Coupling Coefficient Plot (collapsed)
Coupling Coefficient Plot (expanded)
Coupling Coefficient Table
Coupling Coefficient by Layer

1. Click

   Net Length Summary.

   A summary table for enabled signal nets is displayed.

Signal Net Name	Net Length (mm)
FCHIP_A1	2785.115
FCHIP_A2	3663.313
FCHIP_A3	4550.800
FCHIP_A6	3442.505
FCHIP_A7	2442.132
FCHIP_A12	2484.810
FCHIP_A13	3407.297
FCHIP_A16	4623.287
FCHIP_A17	3678.313
FCHIP_A18	2785.115
FCHIP_B3	2865.140
FCHIP_B4	3758.338
FCHIP_B5	4659.820
FCHIP_B6	3613.363
FCHIP_B13	3671.353
FCHIP_B14	4651.536
FCHIP_B15	3758.338
FCHIP_B16	2865.140
FCHIP_C2	3758.338
FCHIP_C17	3783.338
FCHIP_F2	3588.363
FCHIP_F17	3563.363
FCHIP_H2	2639.099
FCHIP_H17	2639.099
FCHIP_K2	3729.343
FCHIP_K17	3654.784
FCHIP_M2	2748.339
FCHIP_M17	2699.454
FCHIP_U3	3640.140
FCHIP_U5	3638.069
FCHIP_U6	2846.734
FCHIP_U13	2753.008

2. Click:

Net Impedance Summary

A summary table is displayed. The table includes the following information for all selected Signal Nets.

- Net Name
- Number of Tace Reference Discontinuity
- Number of Vias
- Maximum Impedance
- Minimum Impedance
- Dominant Impedance
- Dominant Impedance Length Percentage
- Trace / Wirebond Length

Net Count	Net Name	No. of Trace Ref...	No. of Vias	Maximum Impedance (Ohm)	Minimum Impedance (Ohm)	Dominant Impedance (Ohm)	Dominant Imp Length (%)	Trace _wirebond total length(m
1	Net_1	0	2	74.012	35.423	74.012	92.567	6.727
2	Net_2	0	2	74.012	35.423	74.012	91.363	5.790
3	Net_3	0	2	74.012	35.423	74.012	91.562	5.927
4	Net_4	0	2	74.012	35.423	74.012	92.618	6.774
5	Net_5	0	2	74.012	35.423	74.012	93.888	8.181
6	Net_6	0	2	74.012	35.423	74.012	92.951	7.094
7	Net_7	0	2	74.012	35.423	74.012	92.412	6.590
8	Net_8	0	2	74.012	35.423	74.012	91.860	6.143
9	Net_9	0	2	74.012	35.423	74.012	91.255	5.718
10	Net_10	0	2	74.012	35.423	74.012	90.623	5.333
11	Net_11	0	2	74.012	35.423	74.012	89.975	4.988
12	Net_12	0	2	74.012	35.423	74.012	89.315	4.680
13	Net_13	0	2	74.012	35.423	74.012	88.811	4.469
14	Net_14	0	2	74.012	35.423	74.012	88.227	4.247

3. Click:

Net Coupling Summary

A summary table is displayed. It includes the following information for all selected Signal Nets.

- Net Name
- Aggressor Net with Max Coupling
- Maximum Coupling Coefficient
- % Length with Max Coupling over the total length
- % Length with Coupling Coefficient > 0.05
- % Length with Coupling Coefficient 0.03 - 0.05
- % Length with Coupling Coefficient 0.02 - 0.03

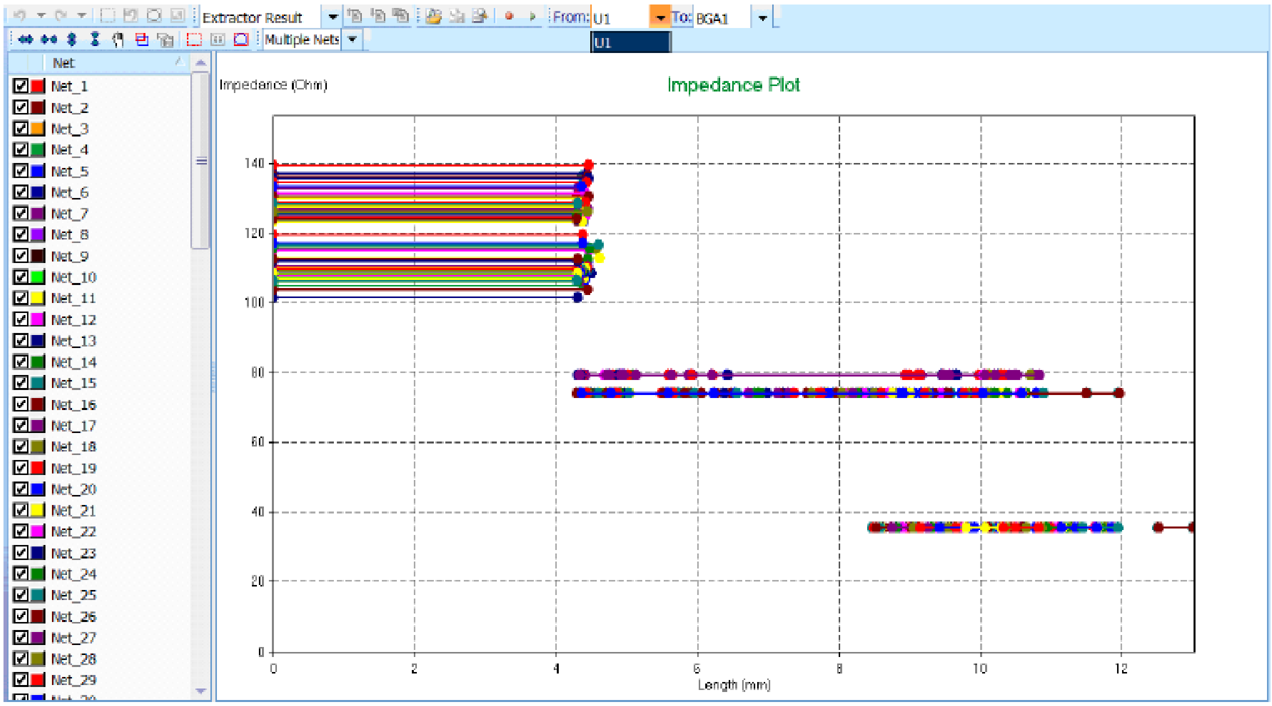
Net count	Net name	Aggressor net with max coupling	Max coupling coefficient	% length with max coupling	% length with coupling coef > 0.05	% length with coupling coef 0.03~0.05	% length with
1	**Net_1**	Net_10	**0.109**	39.193	**39.193**		
2	**Net_2**	Net_13	**0.106**	43.369	**43.369**		
3	**Net_3**	Net_18	**0.042**	42.764		42.764	
4	**Net_4**	Net_21	**0.140**	39.293	**80.033**		
5	**Net_5**	Net_24	**0.057**	34.417	**34.417**		
6	**Net_6**	Net_7	**0.036**	37.776		37.776	
7	**Net_7**	Net_8	**0.057**	39.445	**39.445**		
8	**Net_8**	Net_9	**0.110**	41.265	**41.265**		
9	**Net_9**	Net_8	**0.111**	42.986	**42.986**		
10	**Net_10**	Net_1	**0.109**	44.843	**44.843**		
11	**Net_11**	Net_12	**0.043**	46.771		46.771	
12	**Net_12**	Net_2	**0.044**	48.573		48.573	
13	**Net_13**	Net_2	**0.107**	49.801	**49.801**		

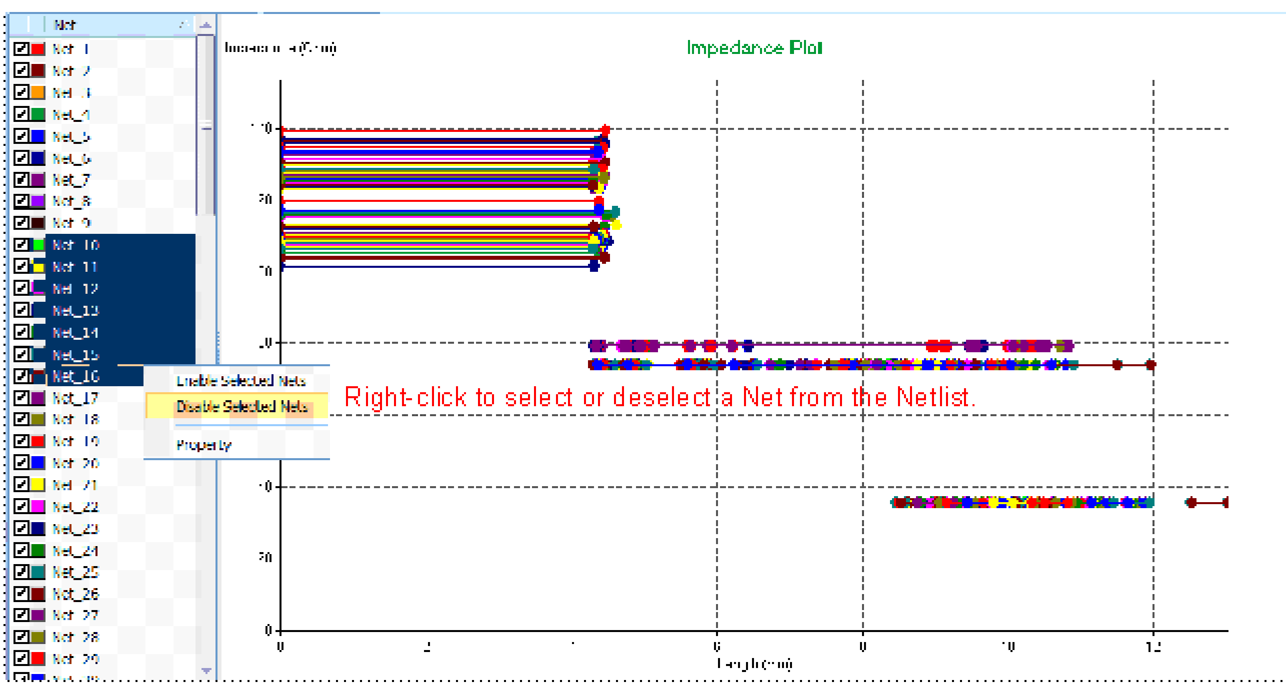
4. Click:

Impedance Plot (collapsed)

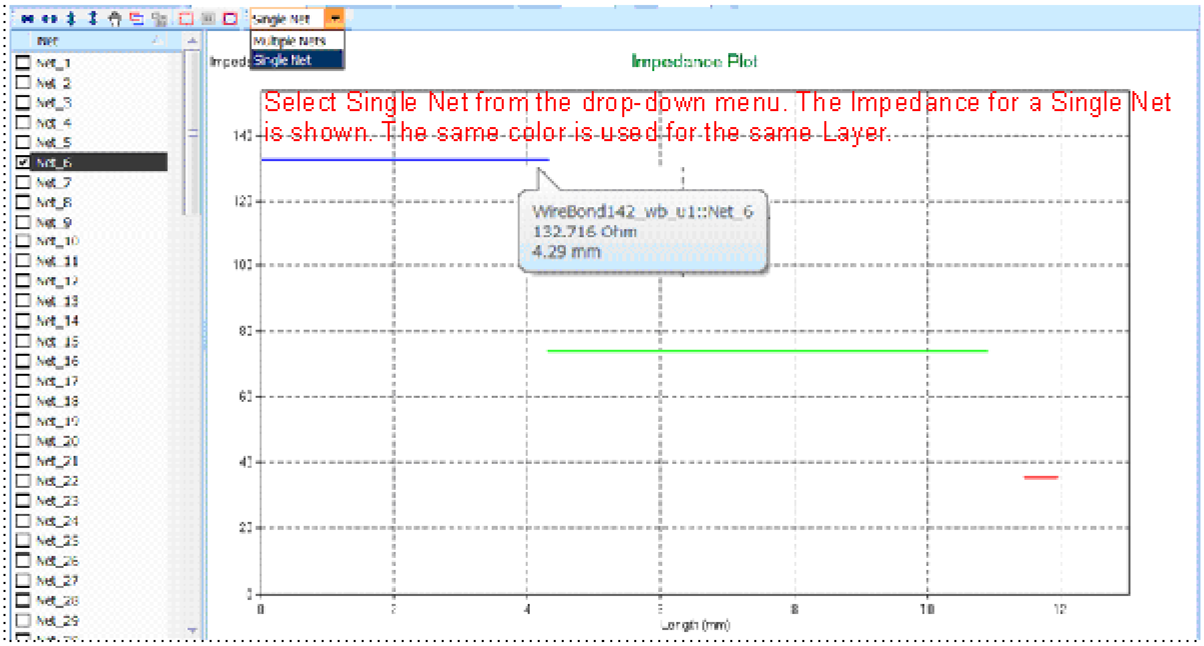
The **Trace and Wirebond Impedance** is displayed (collapsed) from Die to Board Circuit.

NOTE!	If it is a Multi-die package, then follow these steps. 1. Select a Component Name in the From: drop-down list. 2. Select a Component Name in the To: drop-down list. The Trace and Wirebond Impedance along the path of the Start-Component to the End-Component is shown for all Nets between the selected two components.





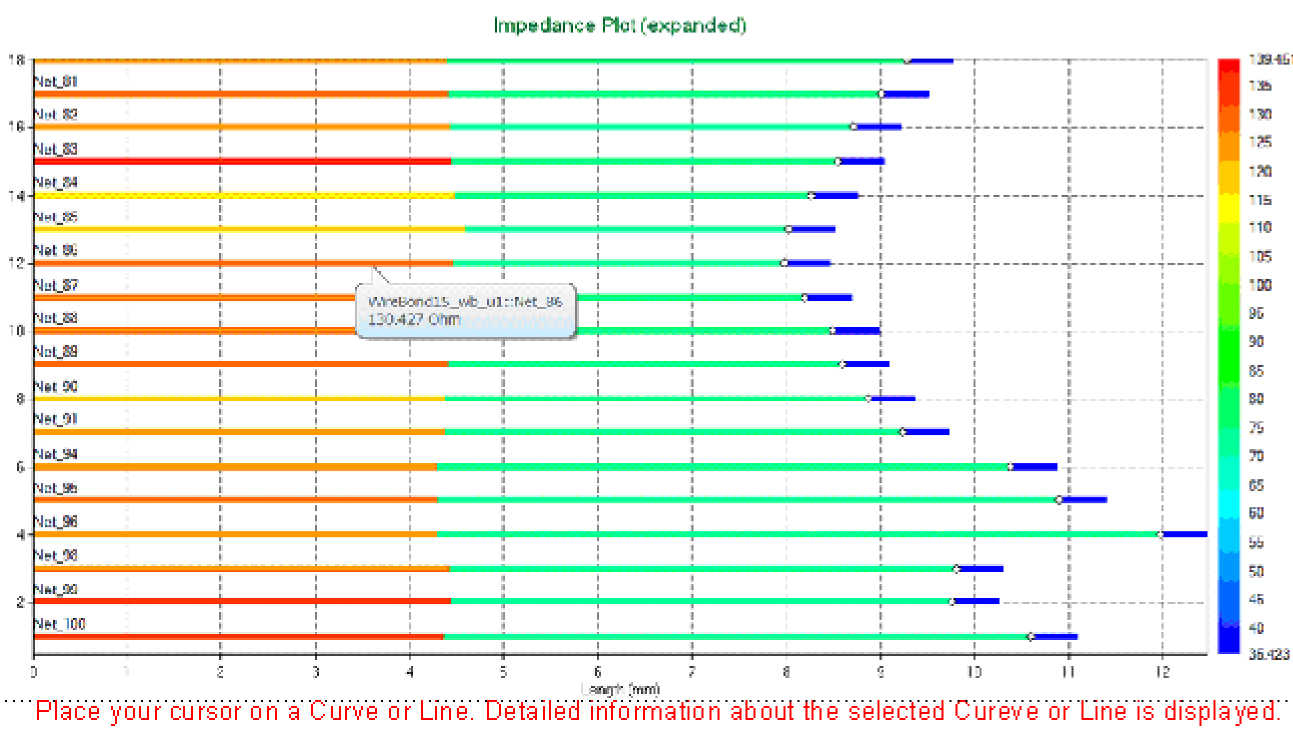
If there are any reference changes on the same layer, those changes are clearly marked.



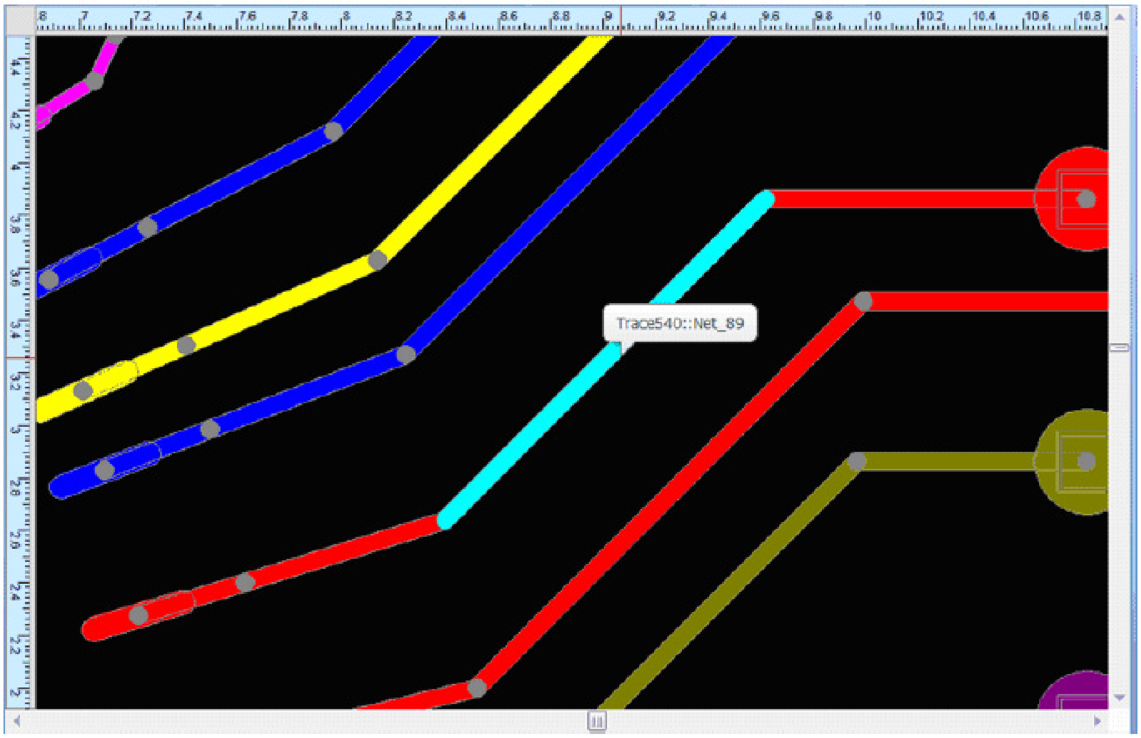
5. Click:

Impedance Plot (expanded)

Trace and Wirebond Impedance is displayed from Die to Board Circuit (expanded).



NOTE!	To use the Cross Probe feature, double-click on any Trace section in the plot. The Trace is shown in the layout with the highlighted line.



Impedance

1. Click:

   Impedance Table

   For each Net, the table displays each:

   - Section name
   - Length percentage over the whole length
   - Impedance
   - Total length
   - Distance from starting component

Net
Net_1
Net_2
Net_3
Net_4
Net_5
Net_6
Net_7
Net_8
Net_9
Net_10

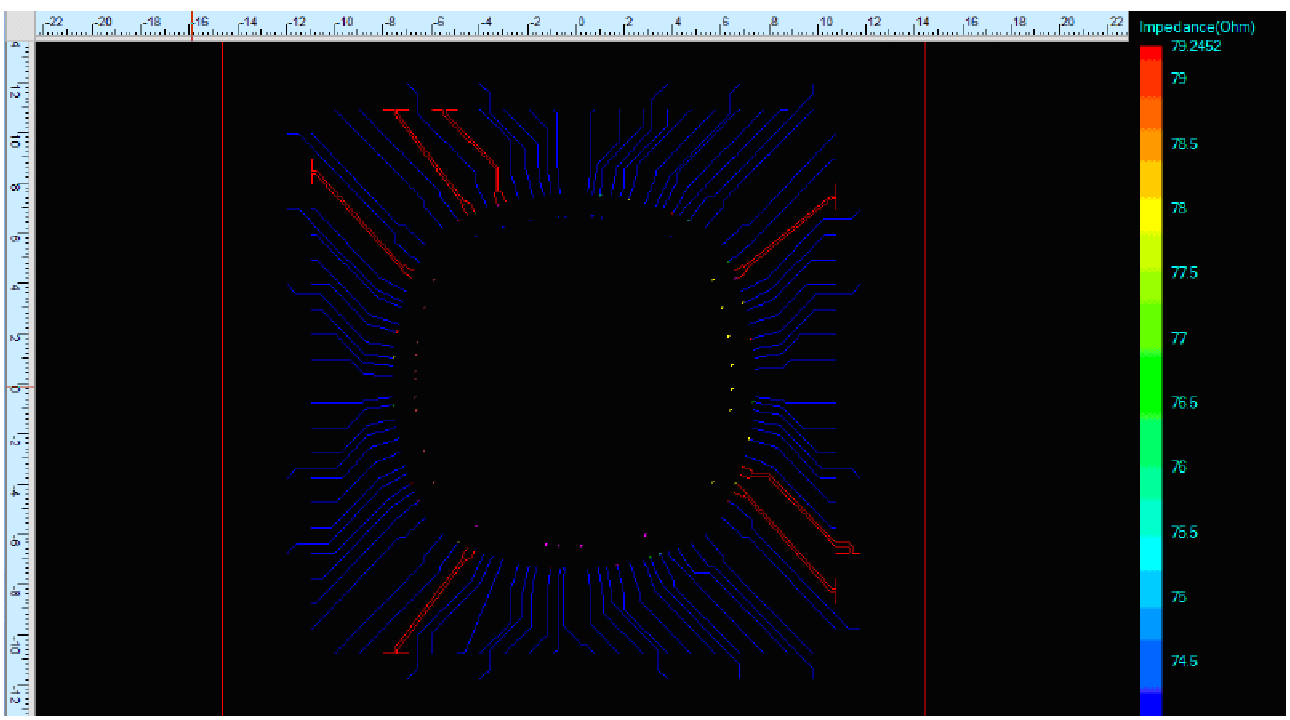
Trace/wirebond Name	% of its net length	Impedance(Ohm)	Length(mm)	Trace/wirebond distance from Starting Component(mm)
WireBond175_wb_u...	37.32%	106.885	4.336	0.000
Trace701::Net_1	3.68%	74.012	0.427	4.336
Trace700::Net_1	6.89%	74.012	0.800	4.763
Trace699::Net_1	27.08%	74.012	3.146	5.563
Trace698::Net_1	11.24%	74.012	1.306	8.710
Trace697::Net_1	4.72%	74.012	0.548	10.015
Trace239::Net_1	4.30%	35.423	0.500	11.118

2. Click:

   Impedance in Layout

3. Select a Component Name in the **From:** drop-down list.

4. Select a Component Name in the **To:** drop-down list. The Impedance along the Traces and Wire-bonds between the two selected components is shown.



Coupling Coefficient

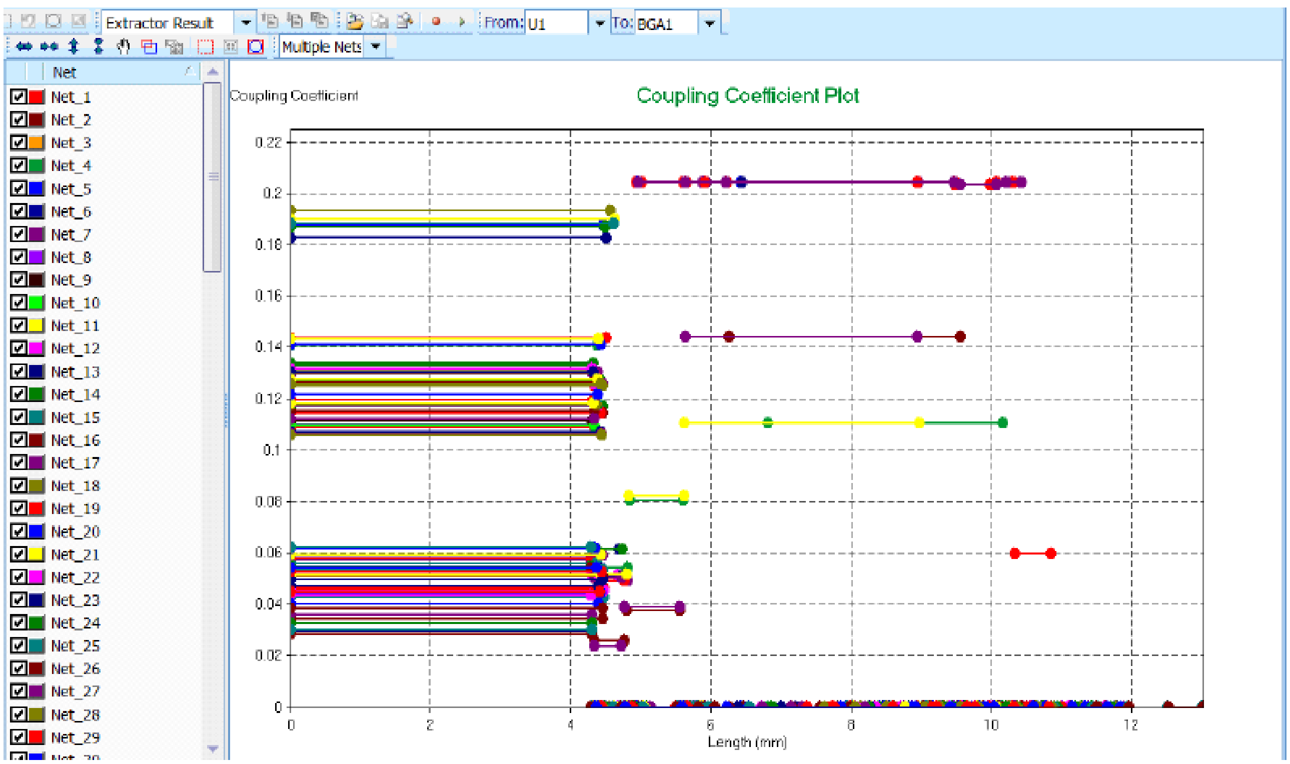
1. Click:

   Coupling Coefficient Plot (collapsed)

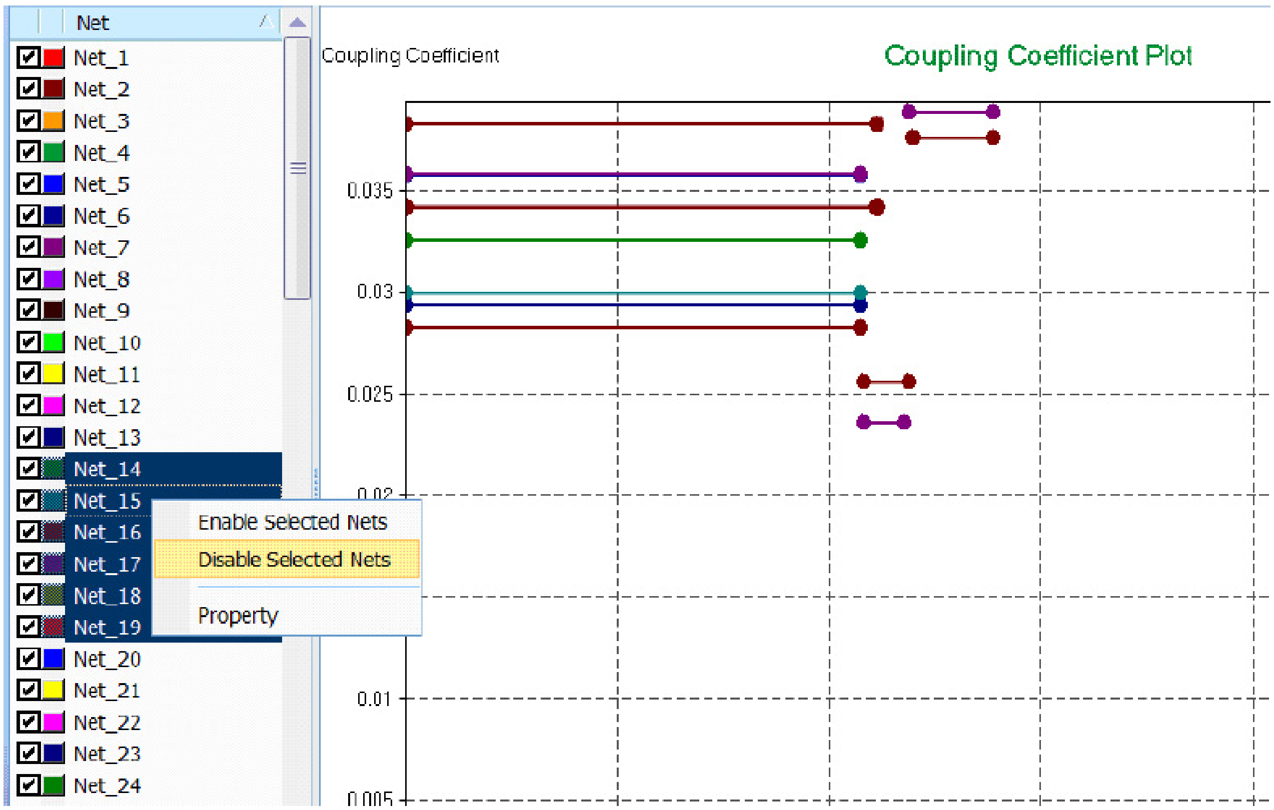
2. Select a Component Name in the **From:** drop-down list.

3. Select a Component Name in the **To:** drop-down list.

   The strongest coupling coefficient of each Trace and Wirebond section is displayed (collapsed) from the **Start Component** to the **End Component**.



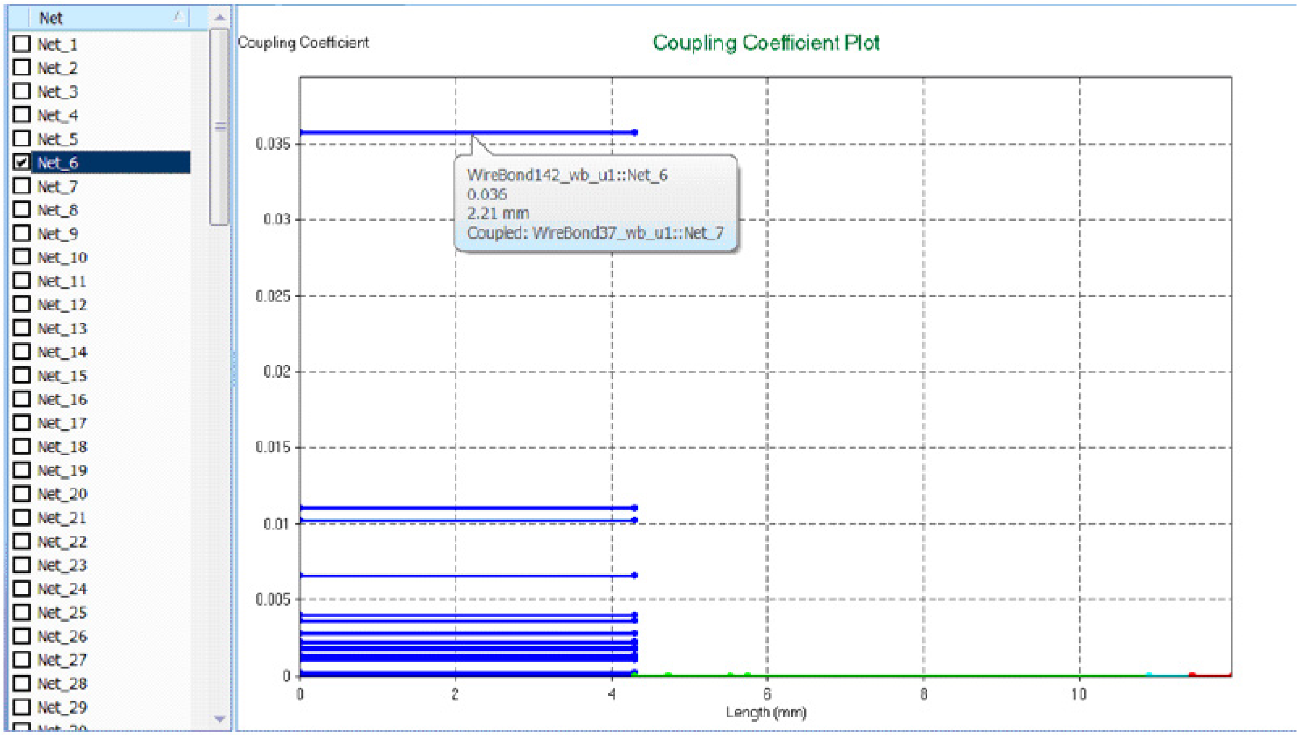
4. Select Nets in the Netlist.
5. Right-click to select or deselect the Net.



6. Select **Single Net** from the top drop-down menu. A **Single Net Coupling Coefficient** is displayed. It includes a few of the strongest couplings to each section.

   The same color is used for the same layer.

7. Place your mouse on a Curve. Detailed information about the Curve is displayed.



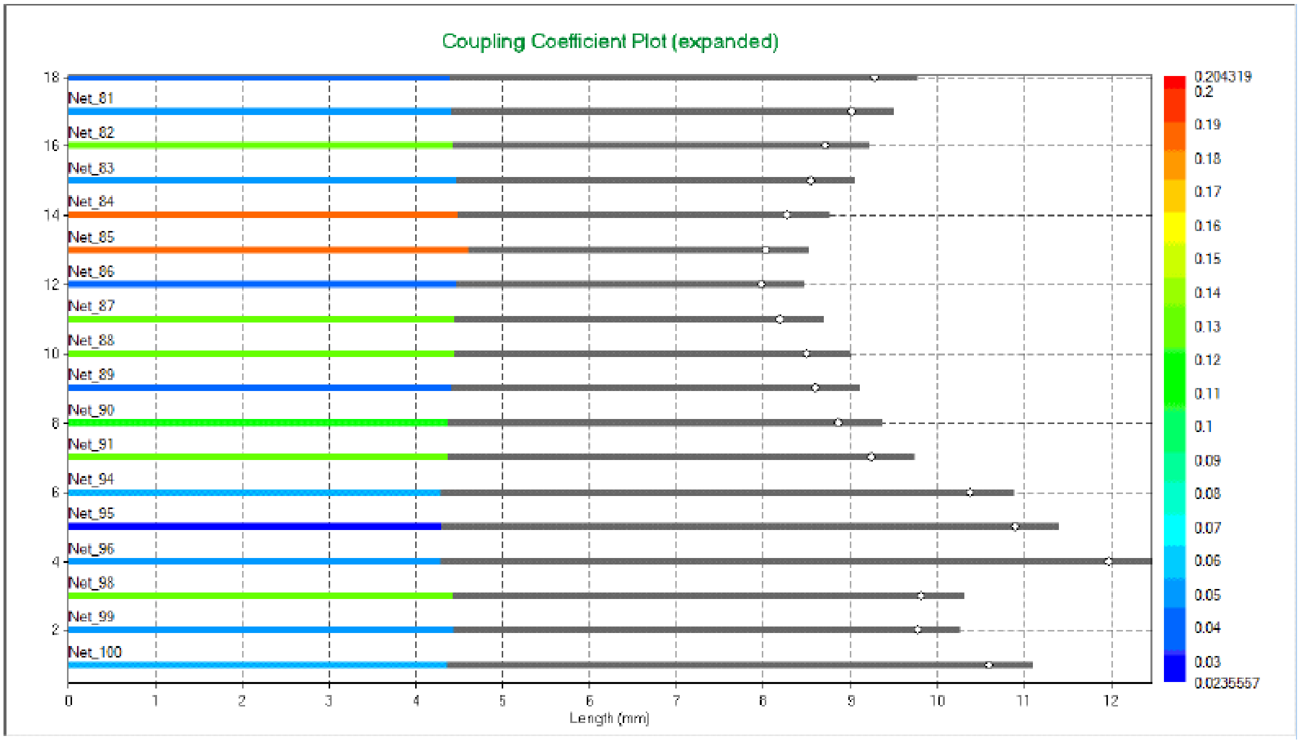
8. Click:

Coupling Coefficient Plot (expanded)

9. Select a Component Name in the **From:** drop-down list.

10. Select a Component Name in the **To:** drop-down list.

The strongest **Coupling Coefficient** for each Trace and Wirebond section is displayed (expanded) from the **Start Component** to the **End Component**.



11. Click:

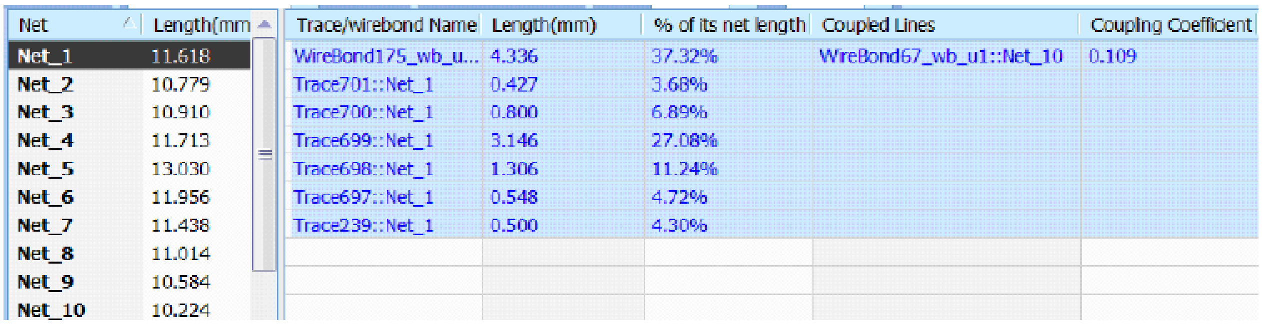
Coupling Coefficient Table

12. Select a Component Name in the **From:** drop-down list.

13. Select a Component Name in the **To:** drop-down list.

The strongest Coupling Coefficient for each Trace and Wirebond section is listed with the table.

14. Choose a specific Net to view the detailed results.



Net	Length(mm
Net_1	11.618
Net_2	10.779
Net_3	10.910
Net_4	11.713
Net_5	13.030
Net_6	11.956
Net_7	11.438
Net_8	11.014
Net_9	10.584
Net_10	10.224

Trace/wirebond Name	Length(mm)	% of its net length	Coupled Lines	Coupling Coefficient
WireBond175_wb_u...	4.336	37.32%	WireBond67_wb_u1::Net_10	0.109
Trace701::Net_1	0.427	3.68%		
Trace700::Net_1	0.800	6.89%		
Trace699::Net_1	3.146	27.08%		
Trace698::Net_1	1.306	11.24%		
Trace697::Net_1	0.548	4.72%		
Trace239::Net_1	0.500	4.30%		

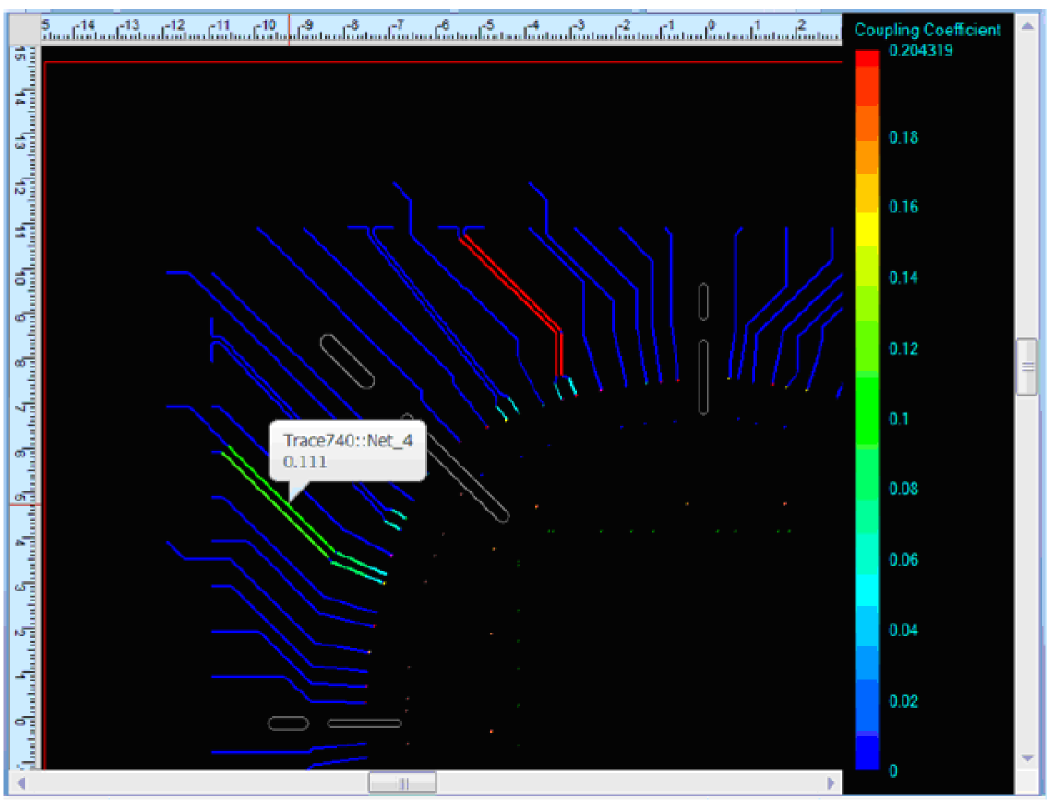
15. Click:

    Coupling Coefficient in Layout

16. Select a Component Name in the **From:** drop-down list.
17. Select a Component Name in the **To:** drop-down list.

    The strongest Coupling Coefficient of each Trace and Wirebond section is overlaid on the Layout.

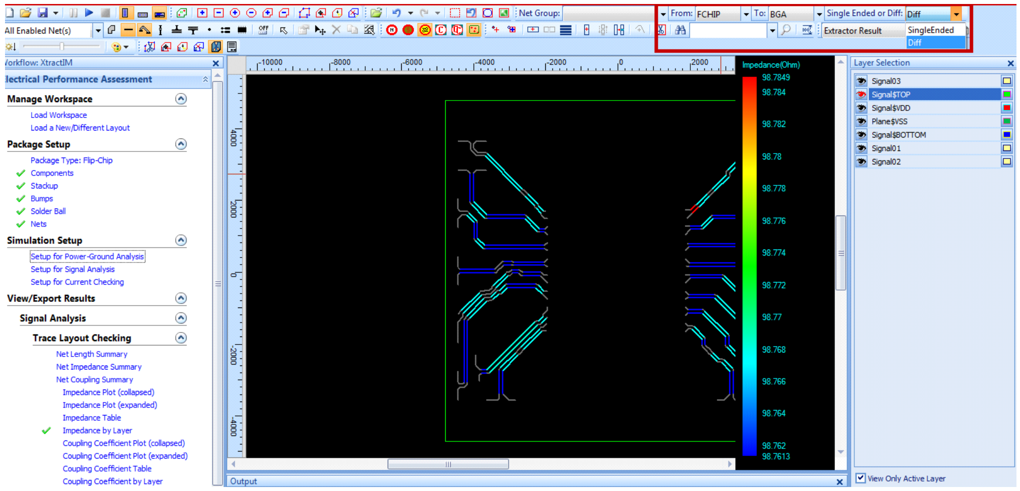
18. Move your cursor onto the Layout. The **Net Name and Coupling Coefficient i**s shown.



Single-Ended and Diff. Pair Net Checking Switching

If there are differential pair signal nets defined and enabled in Net Manager, you can switch the trace checking report and plot between single-ended and differential pair signal nets.

The impedance and coupling coefficient table and plot can be switched by choosing **SingleEnded** or **Diff** from the **Single Ended or Diff** column.

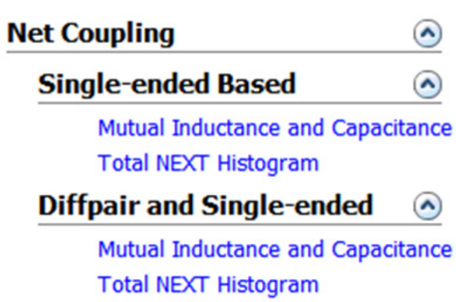


Net Couplings

This feature is for Single-die Single-BGA package only. Using all Power and Ground Nets ad Reference Nets, couplings report the:

- Mutual Inductance
- Mutual Capacitance
- Crosstalk among all Signal Nets

Every signal net can be considered as a single-ended net. Or, if differential pair exists, it can be considered as a whole in discussing crosstalk with other nets, i.e. Differential pair vs. Single-ended or another differential pair.



Signle-ended Based

1. Click:

Mutual Inductance and Capacitance (under Signle-edned Based)

A table shows:

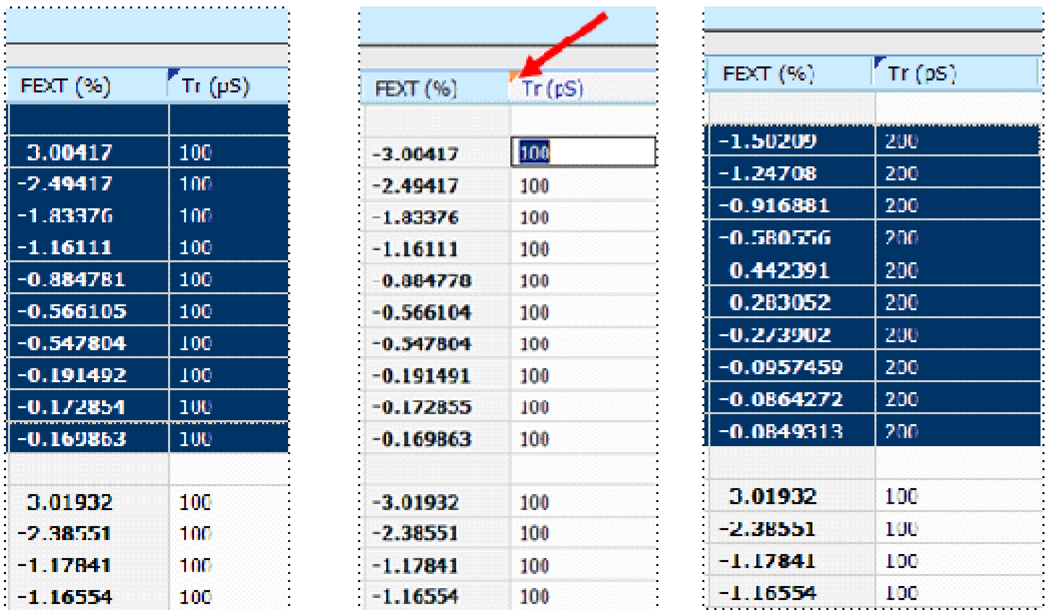
- Resistance
- Mutual Inductance and Capacitance
- Near-ended Crosstalk
- Total Near-ended Crosstalk of each Net as Victim
- Far-ended Crosstalk assuming the Rise Rime is 100 ps

Net i	Net j	Rij (mOhm)	Lij (nH)	Cij (pF)	NEXT (%)	Total NEXT (%)	FEXT (%)	Tr (pS)
Net_1	**Net_1**	271.466	**6.77129**	1.40194		8.56234		
Net_1	**Net_10**	0	**0.673844**	0.0543725	**3.63607**		-**3.00417**	100
Net_1	**Net_9**	0	**0.433952**	0.0181933	**1.99271**		-**2.49417**	100
Net_1	**Net_8**	0	**0.254391**	0	**0.958141**		-**1.83376**	100
Net_1	**Net_11**	0	**0.160449**	0	**0.629086**		-**1.16111**	100
Net_1	**Net_7**	0	**0.123555**	0	**0.456147**		-**0.884778**	100
Net_1	**Net_12**	0	**0.0781773**	0	**0.310549**		-**0.566104**	100
Net_1	**Net_6**	0	**0.076406**	0	**0.280166**		-**0.547804**	100
Net_1	**Net_14**	0	**0.0263146**	0	**0.10684**		-**0.191491**	100
Net_1	**Net_16**	0	**0.0235187**	0	**0.096987**		-**0.172855**	100
Net_1	**Net_15**	0	**0.0232635**	0	**0.0956424**		-**0.169863**	100
Net_2	**Net_2**	257.394	**6.36301**	1.3223		8.51554		
Net_2	**Net_13**	0	**0.671347**	0.0534184	**3.83533**		-**3.01932**	100
Net_2	**Net_12**	0	**0.419509**	0.0188933	**2.09137**		-**2.38551**	100
Net_2	**Net_14**	0	**0.161937**	0	**0.679096**		-**1.17841**	100
Net_2	**Net_11**	0	**0.161062**	0	**0.651631**		-**1.16554**	100
Net_2	**Net_15**	0	**0.0884465**	0	**0.375694**		-**0.645807**	100
Net_2	**Net_10**	0	**0.0877581**	0	**0.352181**		-**0.635655**	100
Net_2	**Net_9**	0	**0.0378359**	0	**0.149022**		-**0.272522**	100
Net_2	**Net_16**	0	**0.0315619**	0	**0.13446**		-**0.23197**	100
Net_2	**Net_8**	0	**0.0315379**	0	**0.122435**		-**0.227339**	100
Net_2	**Net_17**	0	**0.0299699**	0	**0.124315**		-**0.217064**	100
Net_3	**Net_3**	259.857	**6.4191**	1.33039		7.15329		
Net_3	**Net_18**	0	**0.371279**	0.0182203	**1.87086**		-**2.05114**	100
Net_3	**Net_19**	0	**0.331794**	0.0120447	**1.58481**		-**2.01174**	100
Net_3	**Net_20**	0	**0.22027**	0	**0.88141**		-**1.59764**	100
Net_3	**Net_17**	0	**0.21053**	0.00616289	**0.991971**		-**1.31208**	100
Net_3	**Net_16**	0	**0.106873**	0	**0.453499**		-**0.78548**	100
Net_3	**Net_21**	0	**0.103712**	0	**0.41142**		-**0.754957**	100

Change Rise Time

Use the following steps to change the Rise Time.

1. Select several lines.
2. Click in the corner of **Ts (ps)**. One cell is enabled for change.
3. Change the enabled cell to a new number.
4. Click **Enter**. All selected lines are changed.



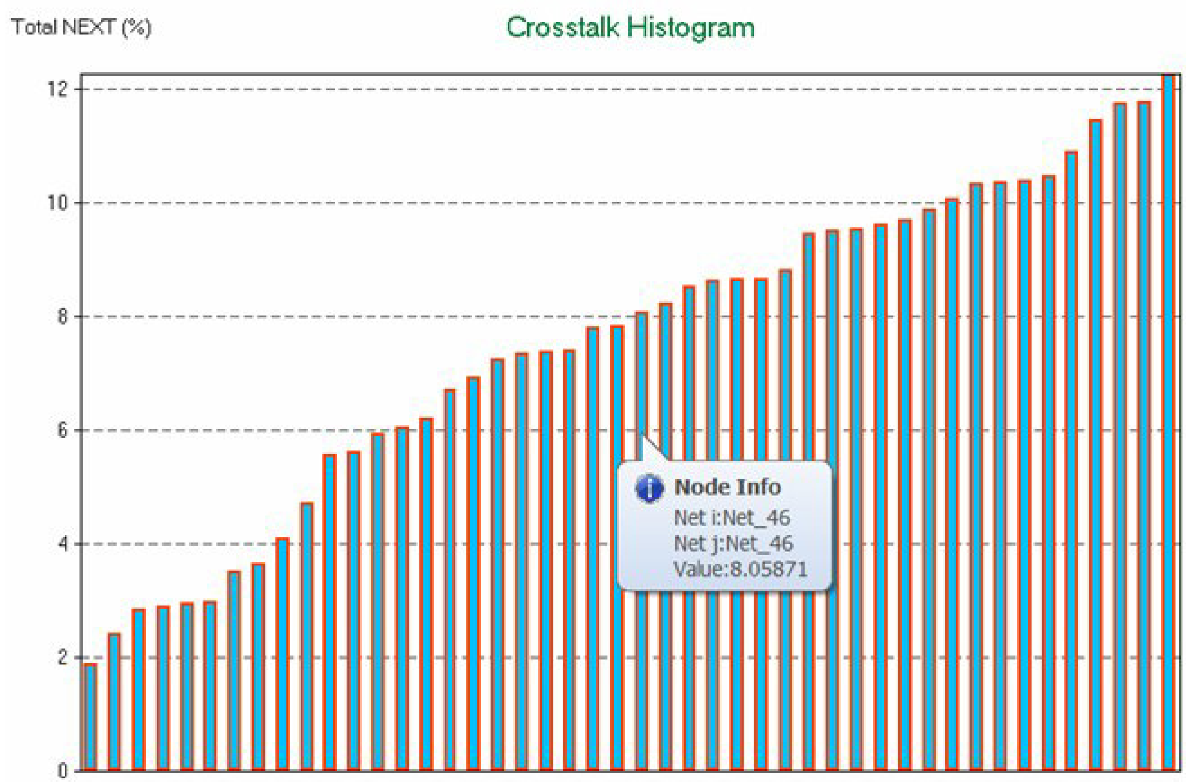
Histogram View

1. Click:

   Total NEXT Histogram View

   The total **NEXT** for each Net as victim is plotted left-to-right and smallest-to-largest.

2. Stop your mouse on a Bar. Detailed information for the Bar is displayed.



Diffpair and Single-ended

1. Click

   Mutual Inductance and Capacitance (Under Diffpair and Single-ended)

   A table shows:

   - Nets: differential pair are shown a whole unit
   - Resistance (*see definition of R for differential pair)*
   - Mutual Inductance and Capacitance
   - Near-ended Crosstalk
   - Total Near-ended Crosstalk of each Net (or differential pair) as Victim
   - Far-ended Crosstalk assuming the Rise Rime is 100 ps

Net i	Net j	Rij (mOhm)	Lij (nH)	Cij (pF)	NEXT (%)	Total NEXT (...	FEXT (%)	Tr (pS)
Diff_L_05N-L_05P	**Diff_L_05N-L_05P**	832.524	**10.8655**	0.583		9.64		
Diff_L_05N-L_05P	**Single_2**	0	**0.7882**	0.097	**5.57**		**1.51**	100
Diff_L_05N-L_05P	**Diff_L_08N-L_08P**	0	**0.4329**	0.126	**3.34**		**0.42**	100
Diff_L_05N-L_05P	**Single_1**	0	**0.1097**	0.006	**0.74**		**0.58**	100
Diff_L_08N-L_08P	**Diff_L_08N-L_08P**	1213.52	**12.6047**	0.971		4.65		
Diff_L_08N-L_08P	**Diff_L_05N-L_05P**	0	**0.4329**	0.126	**3.09**		**-0.07**	100
Diff_L_08N-L_08P	**Single_2**	0	**0.1642**	0.015	**0.79**		**-0.51**	100
Diff_L_08N-L_08P	**Single_1**	0	**0.1180**	0.015	**0.76**		**-0.30**	100
Single_1	**Single_1**	217.127	**3.5739**	0.711		2.30		
Single_1	**Single_2**	0	**0.1343**	0.006	**0.88**		**-0.64**	100
Single_1	**Diff_L_08N-L_08P**	0	**0.1180**	0.015	**0.73**		**-0.25**	100
Single_1	**Diff_L_05N-L_05P**	0	**0.1097**	0.006	**0.68**		**0.51**	100
Single_2	**Single_2**	345.325	**5.5739**	0.942		7.12		
Single_2	**Diff_L_05N-L_05P**	0	**0.7882**	0.097	**5.32**		**1.13**	100
Single_2	**Diff_L_08N-L_08P**	0	**0.1642**	0.015	**0.84**		**-0.61**	100
Single_2	**Single_1**	0	**0.1343**	0.006	**0.96**		**-0.73**	100

Change Rise Time

Use the following steps to change the Rise Time (Similiar to Single-ended Based).

1. Select several lines.
2. Click in the corner of **Ts (ps)**. One cell is enabled for change.
3. Change the enabled cell to a new number.
4. Click **Enter**. All selected lines are changed.

Histogram View

Use the following steps to show the Histogam View (Similar to Single-ended Based).

1. Click:

   Total NEXT Histogram View

   The total **NEXT** for each Net as victim is plotted left-to-right and smallest-to-largest.
2. Stop your mouse on a Bar. Detailed information for the Bar is displayed.

Insertion and Return Loss

Insertion and Loss is reported for all Signal Nets by using Power and Ground Nets as Reference Nets. This feature is only available for Single-die Single-BGA packages.

Insertion and Return Loss

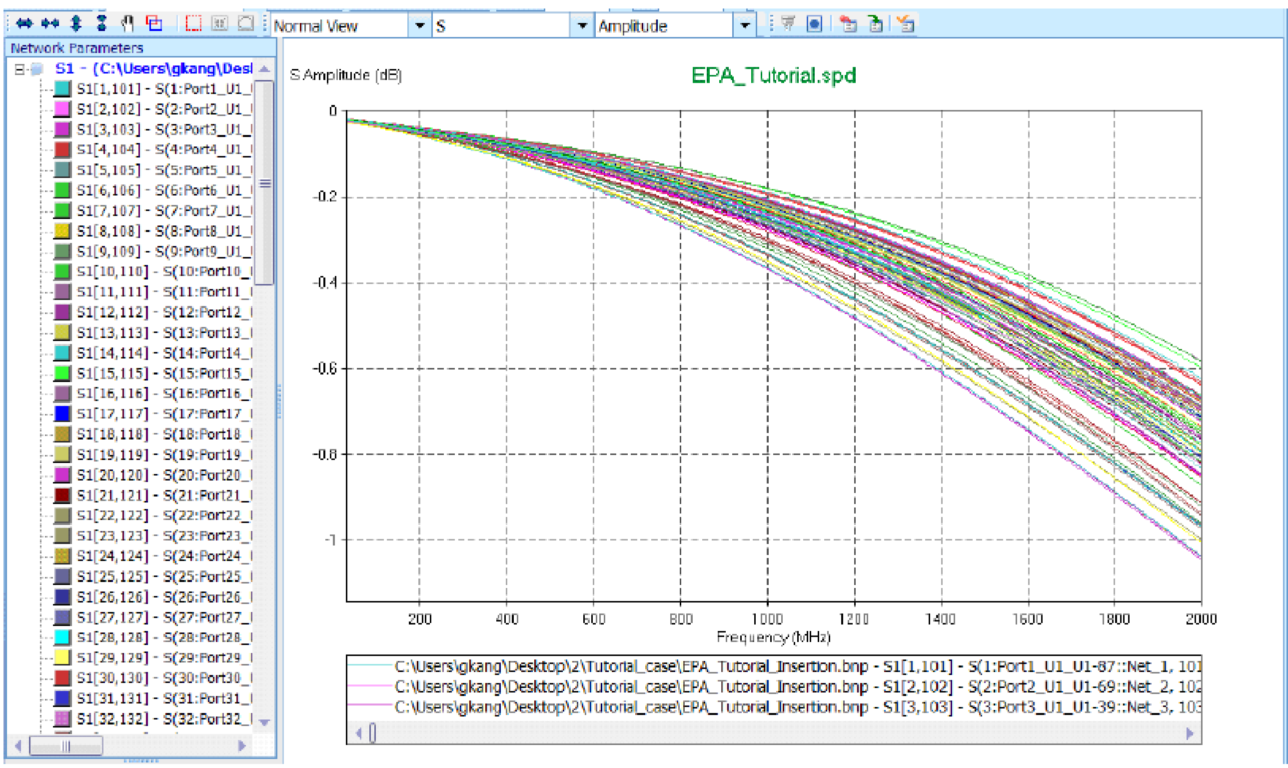
Insertion Loss

Return Loss

1. Click:

   Insertion Loss

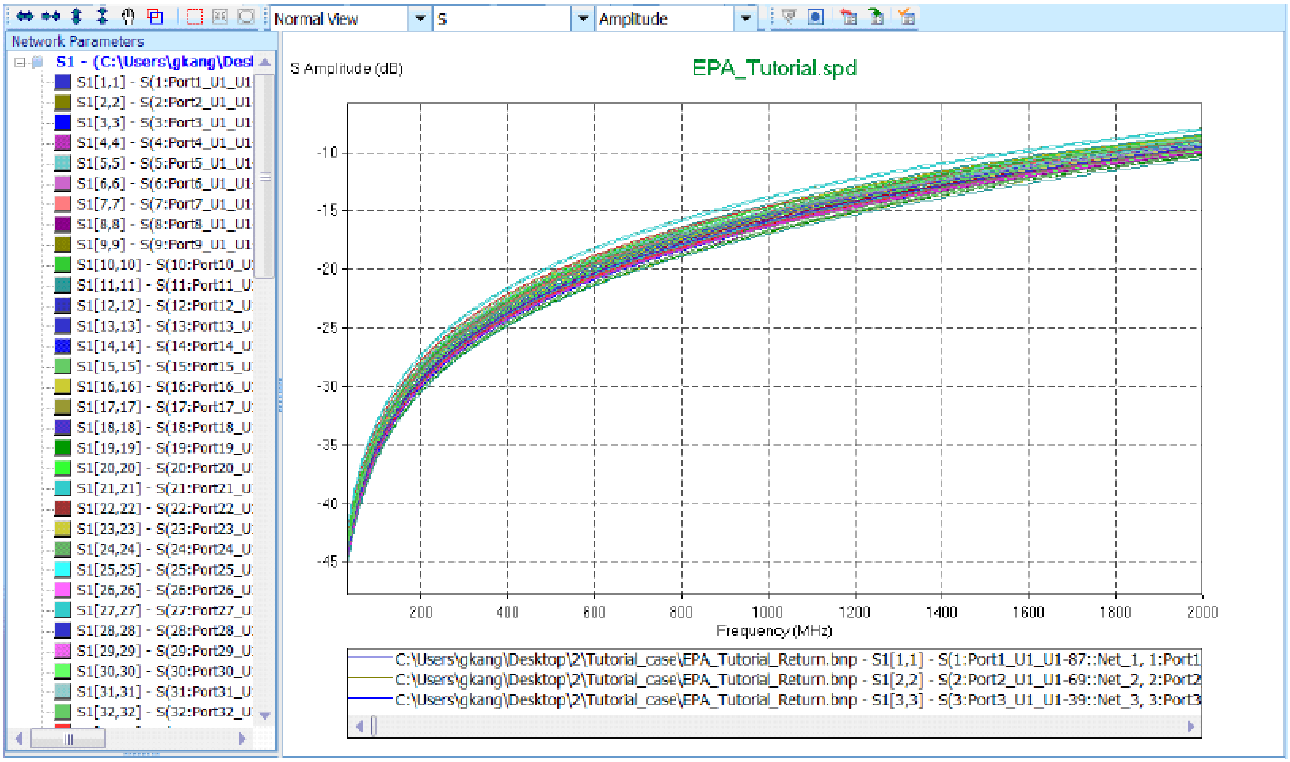
   The Insertion Loss for all Signal Nets is shown.



2. Click:

   Return Loss

   The Return Loss for all Signal Nets is shown.



Current Checking

The Workspace lists options to display results. Click on the desired option.

Current Checking

- Via Counts
- Via Current Histogram
- Via Current by Layer
- Via Current Table
- Voltage Distribution by Layer
- Plane Current Density by Layer

1. Click:

   Via Counts

   The **Via Counts Table** is shown for each Power / Ground Net. The number is classified at each Via starting layer.

Net	Signal$M1->Signal$VSS	Signal$VSS->Signal$VDD	Signal$VDD->Signal$M4
VSS	**28**	256	**256**
VDD_1	**4**	4	**14**
VDD_2	**4**	4	**14**
VDD_3	**4**	4	**14**
VDD_4	**4**	4	**14**

2. Click:

   Via Current Table

   A detailed table showing **Current Density** is displayed. The table includes:

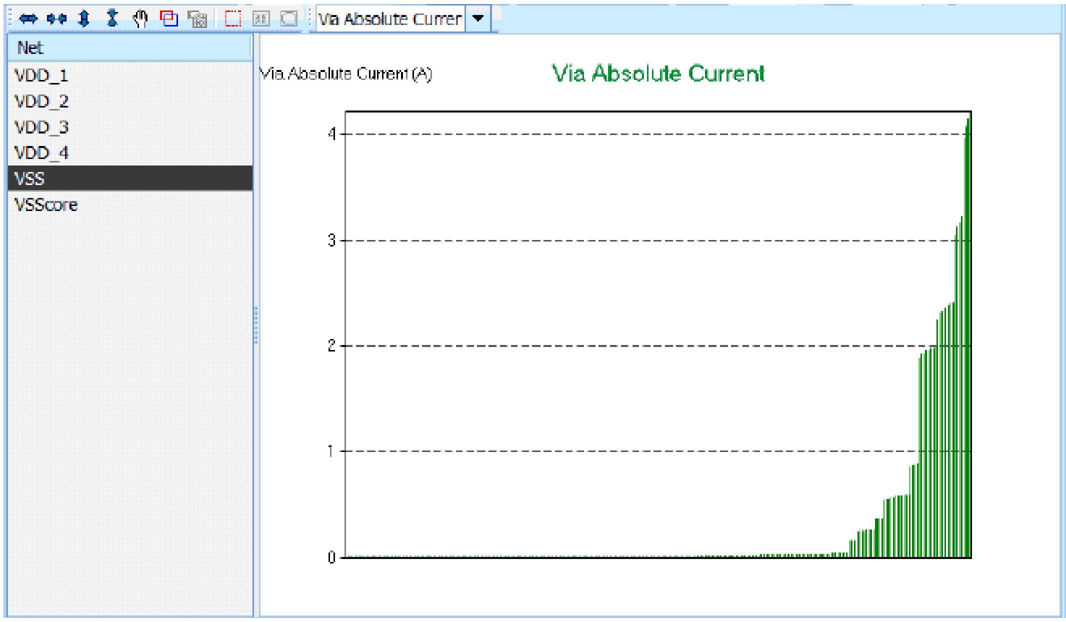
   - Via Name
   - Via Net
   - Actual Current
   - Current Density
   - X,Y Coordinates
   - Start Layer
   - Upper Node Name
   - End Layer
   - Lower Node Name
   - Padstack

Via Name	Net	Current (A)	Current Density (A/mm^2)	PosX (mm)	PosY (mm)	StartLayer	Upper Node	EndLayer	Lower Node	PadStack
Via254::VDD_1	**VDD_1**	-0.0807358	**2.5699**	-13.8536	**-6.8536**	Signal$VDD	**Node2598:...**	Signal$M4	**Node477::...**	via4003
Via264::VDD_1	**VDD_1**	-0.0620086	**1.9738**	-13.8536	**-8.8536**	Signal$VDD	**Node2600:...**	Signal$M4	**Node493::...**	via4003
Via273::VDD_1	**VDD_1**	-0.0372965	**1.18719**	-13.8536	**-10.8536**	Signal$VDD	**Node2602:...**	Signal$M4	**Node507::...**	via4003
Via302::VDD_1	**VDD_1**	-0.0134064	**0.42674**	-13.8536	**-12.8536**	Signal$VDD	**Node2604:...**	Signal$M4	**Node561::...**	via4003
Via334::VDD_1	**VDD_1**	-0.0133563	**0.425144**	-13.8536	**12.8536**	Signal$VDD	**Node2620:...**	Signal$M4	**Node593::...**	via4003
Via343::VDD_1	**VDD_1**	-0.0371082	**1.18119**	-13.8536	**10.8536**	Signal$VDD	**Node2622:...**	Signal$M4	**Node607::...**	via4003
Via370::VDD_1	**VDD_1**	-0.0618354	**1.96828**	-13.8536	**8.8536**	Signal$VDD	**Node2624:...**	Signal$M4	**Node657::...**	via4003
Via378::VDD_1	**VDD_1**	-0.0803316	**2.55703**	-13.8536	**6.8536**	Signal$VDD	**Node2626:...**	Signal$M4	**Node669::...**	via4003
Via388::VDD_1	**VDD_1**	-0.0935236	**2.97695**	-13.8536	**4.8536**	Signal$VDD	**Node2628:...**	Signal$M4	**Node685::...**	via4003

3. Click:

   Via Current Histogram View

   The **Via Absolute Current** window appears.



4. For each Net, the **Via Current** is plotted from the smallest (left) to the largest (right), click:

   Net Name

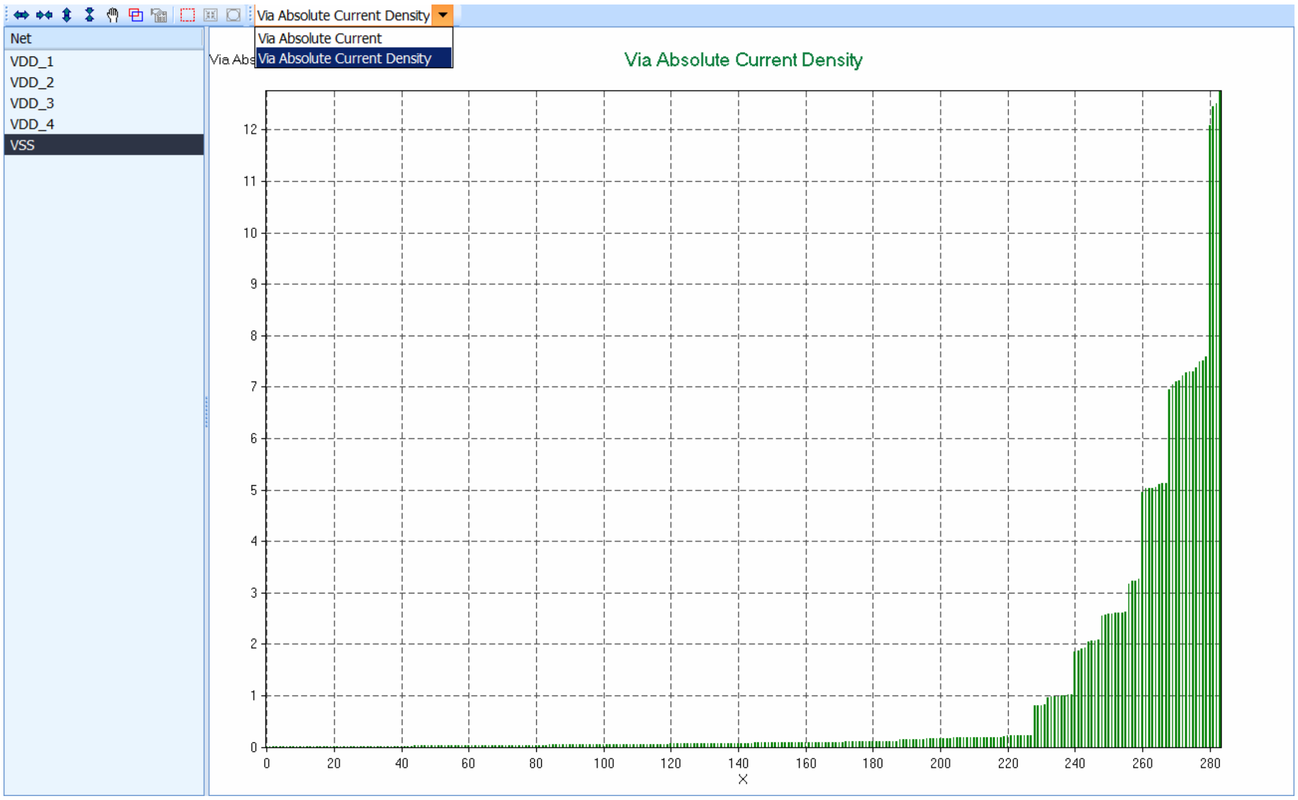
   The **Via Current** is changed.

5. Select **Via Absolute Current Density**.

   It is plotted from the smallest (left) to the largest (right)

6. Click the desired Net Name**.**

   The via current density is changed.



7. Click:

   Current Checking > Via Current in Layout

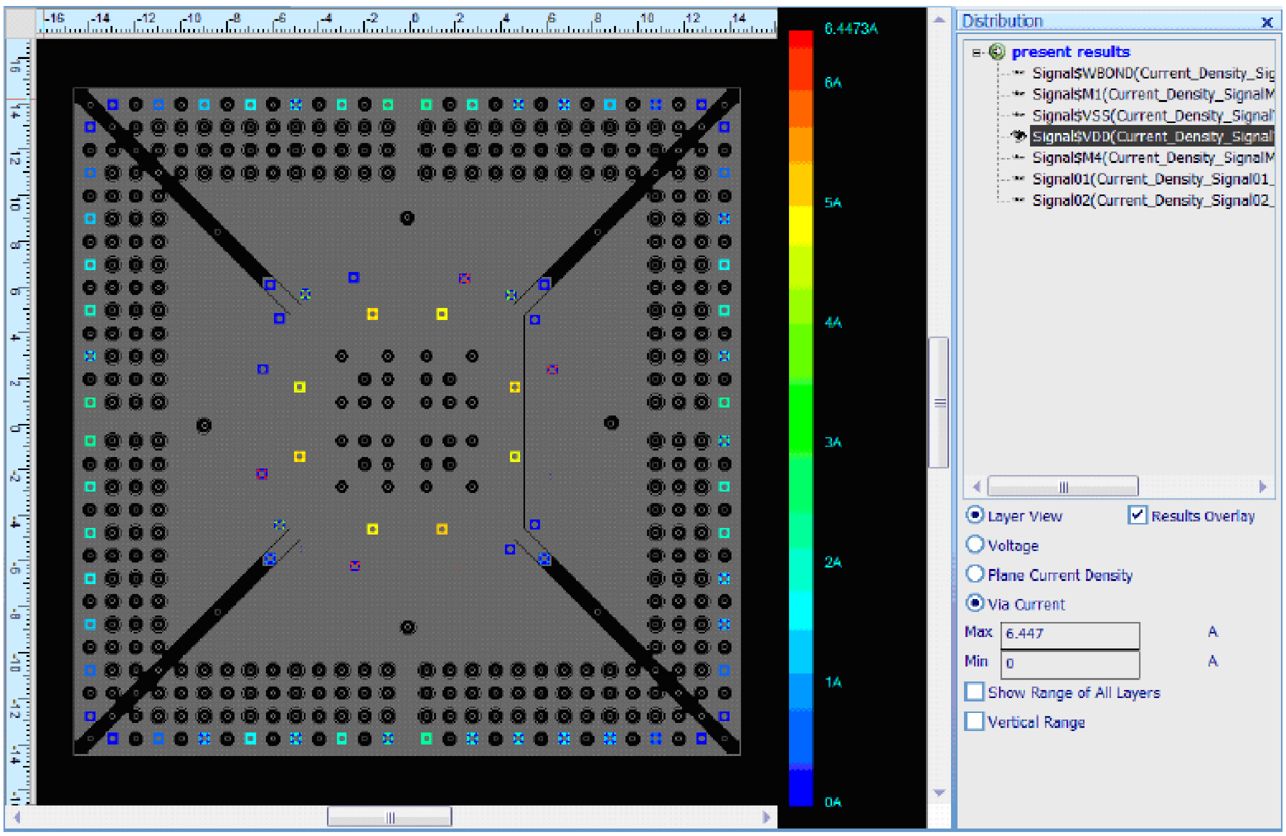
   The **Via Current** is overlaid on the layout to quickly identify the location. The **Color Map** is used as a current value scale.

8. In the **Present Results** window, click on a Layer Name. The Layer changes to reflect the selected Layer Name.

9. Select or Deselect:

   Results Overlay

   The display changes between overlay results and the physical layout.



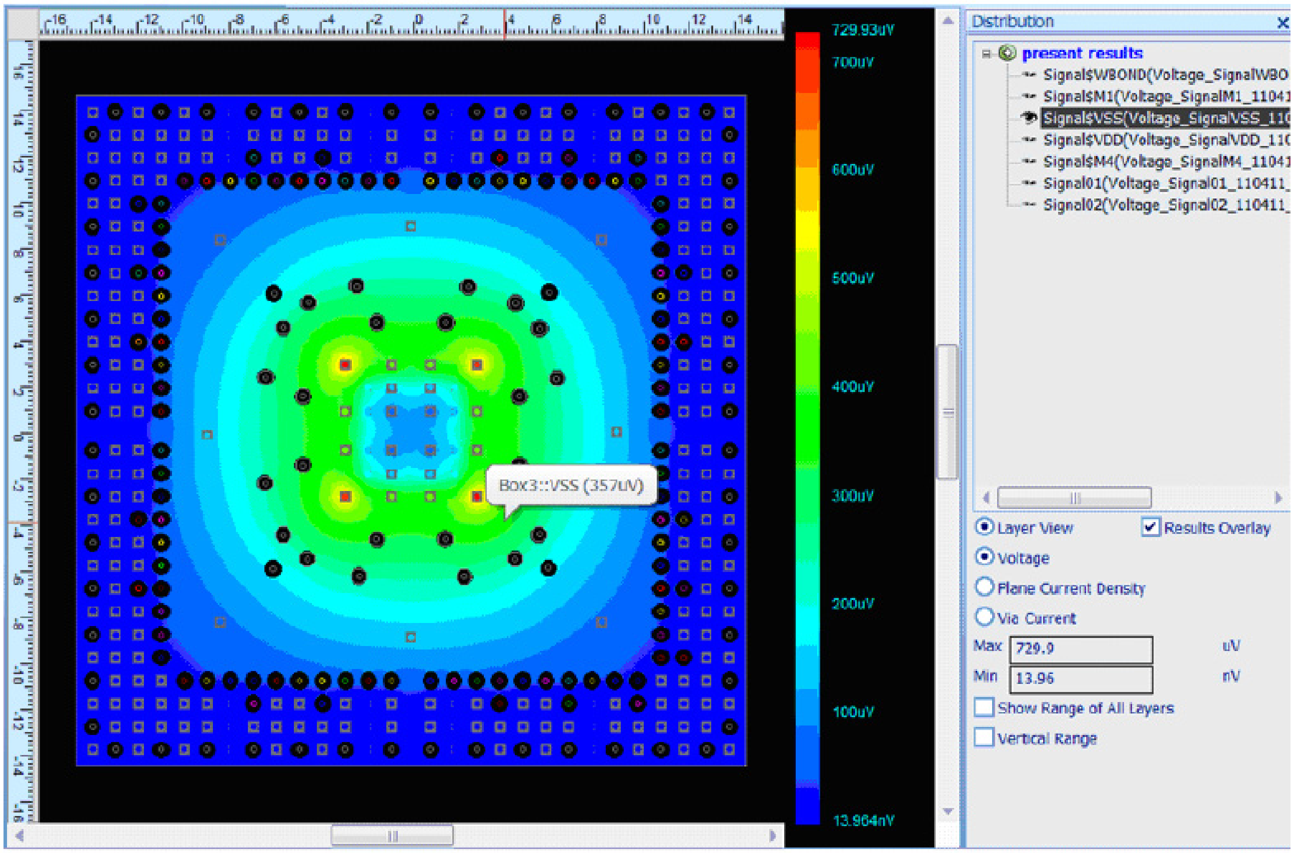
10. Click:

Current Checking > Voltage Distribution in Layout

Voltage distribution (IR Drop) is overlaid for each layer.

11. Select a different layer. The display changes according to the selected layer.

12. Stop the mouse cursor on the display, The actual value is shown.



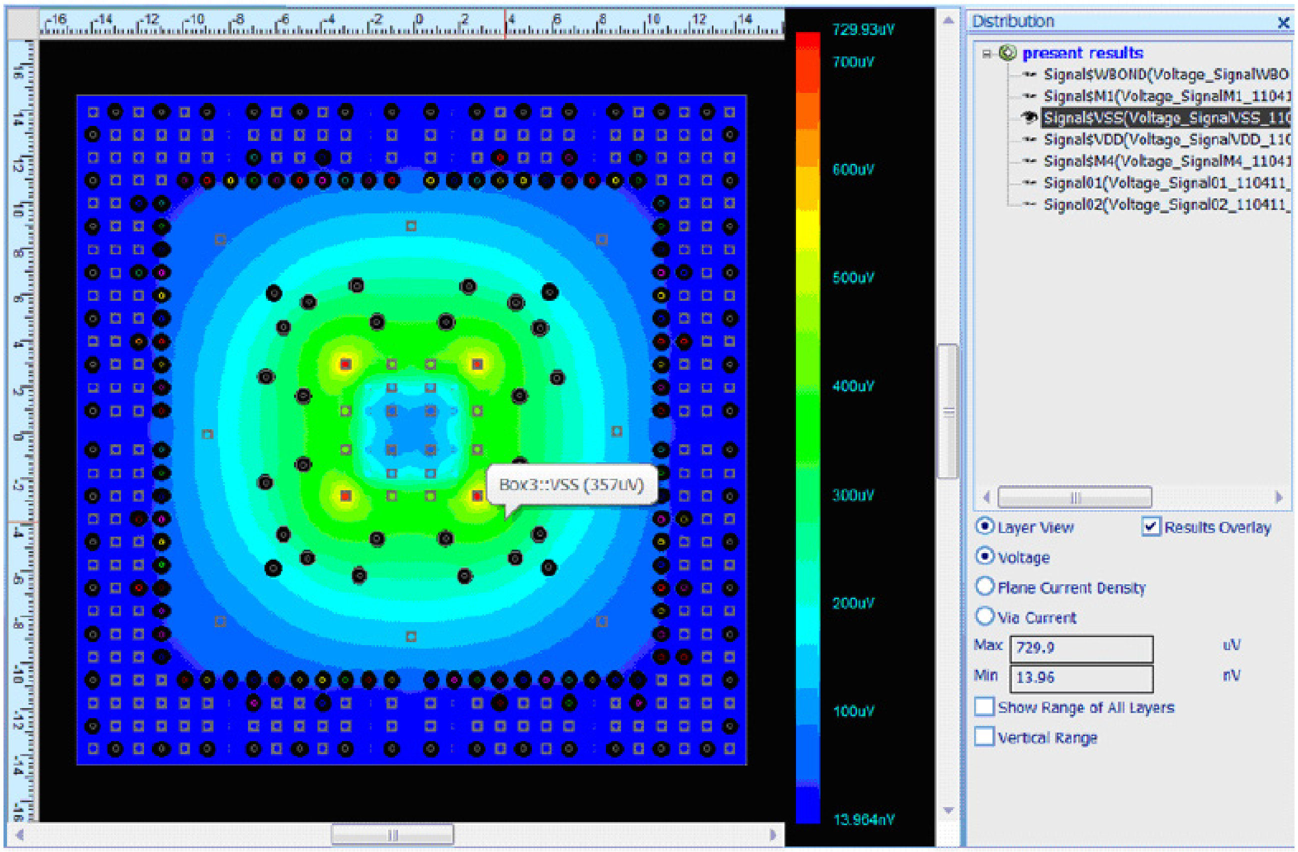
13. Click:

    Current Density Checking > Plane Current Density in Layout

    Each **Plane Current Density** is overlaid on each layer.

14. Select a different layer. The display changes according to the selected layer.

15. Stop the mouse cursor on the display, The actual value is shown.

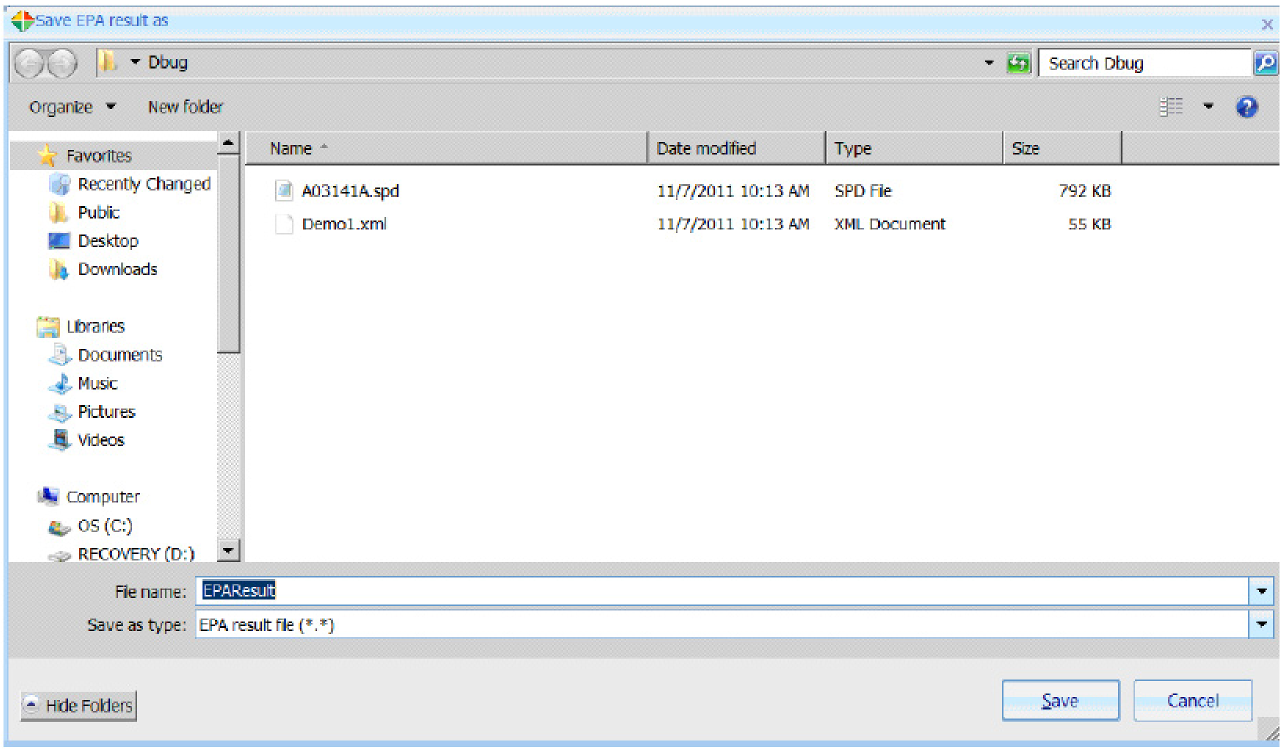


Save Results

1. Click:

   Workspace > Save Results

2. Input a file name.
3. Click **OK** to save the results.

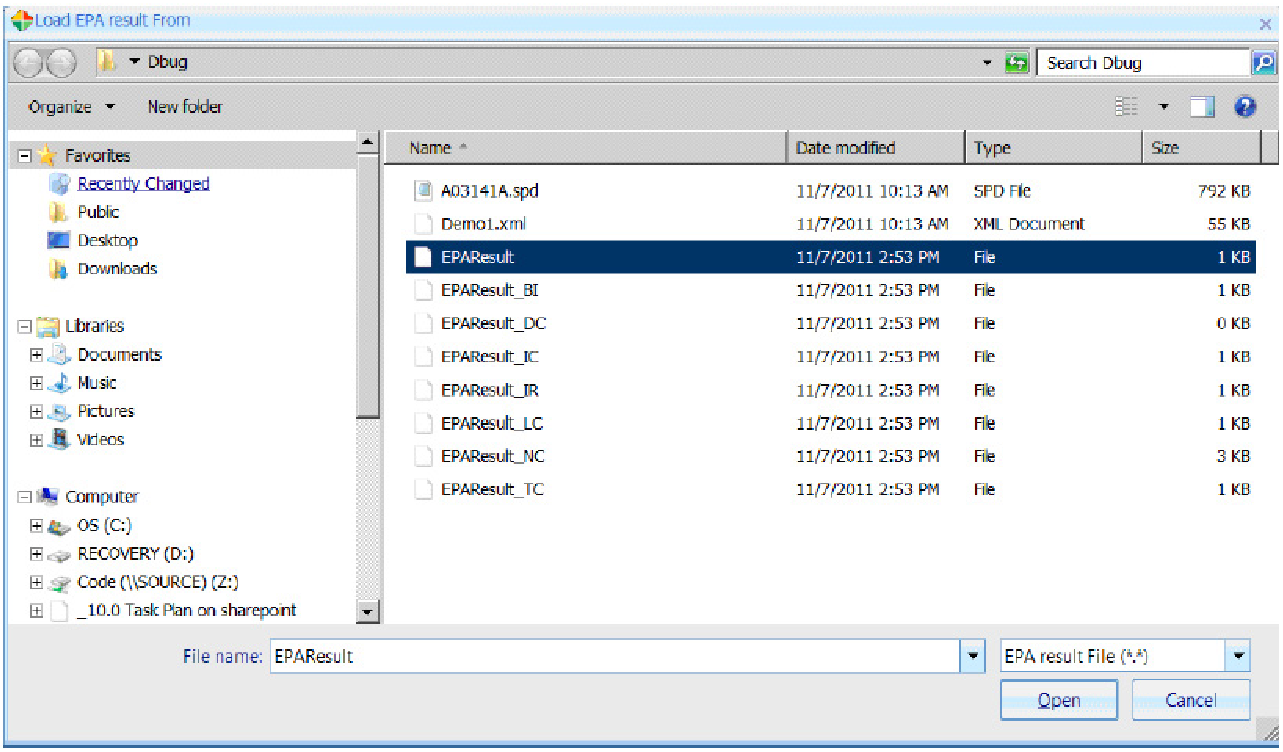


Load Results

1. Launch XtractIM.
2. Load in the Workspace file.
3. Load in the SPD file.
4. Click:

   Workspace > Load Results

   The Load Result window opens.



5. Select the file you want to open.
6. Click **Open** to view the results.

   All results can be displayed with the same operation.

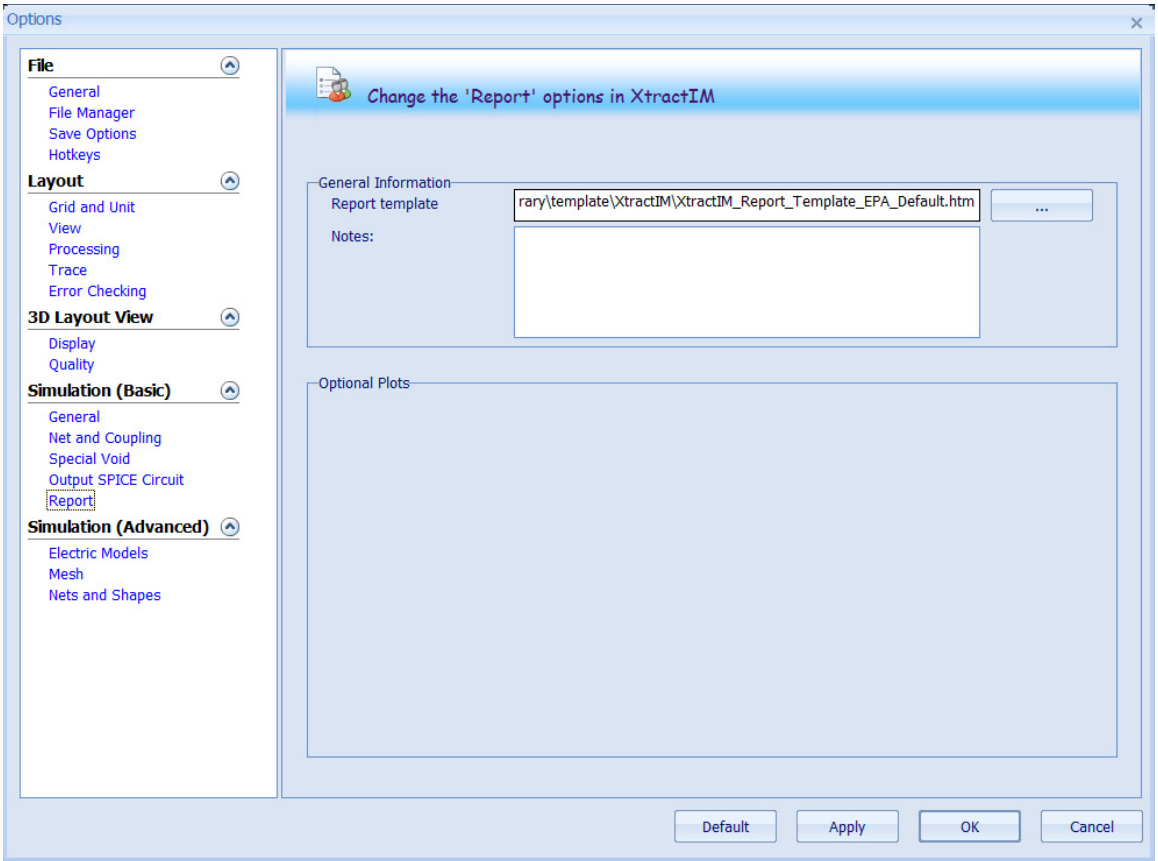
NOTE!	If the XML and SPD file is not loaded before loading the saved results, all overlay features will not work.

Create Report

1. Select

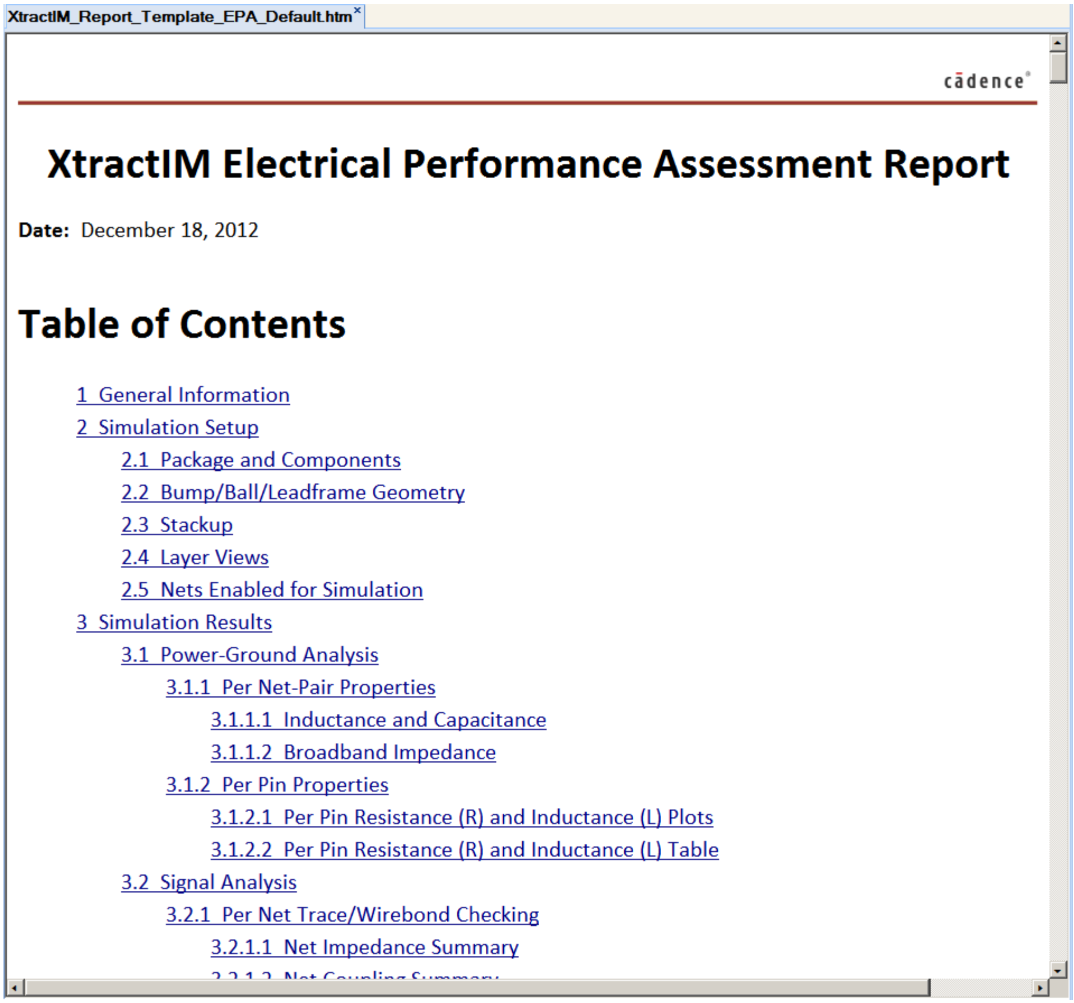
   View / Export Results > Report

   The following window pops up. A default *.htm template is loaded and highlighted.

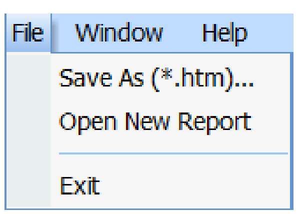

Options
File
General
File Manager
Save Options
Hotkeys
Layout
Grid and Unit
View
Processing
Trace
Error Checking
3D Layout View
Display
Quality
Simulation (Basic)
General
Net and Coupling
Special Void
Output SPICE Circuit
Report
Simulation (Advanced)
Electric Models
Mesh
Nets and Shapes
Change the 'Report' options in XtractIM
General Information
Report template
rary\template\XtractIM\XtractIM_Report_Template_EPA_Default.htm
...
Notes:
Optional Plots
Default
Apply
OK
Cancel

2. Click **OK**.

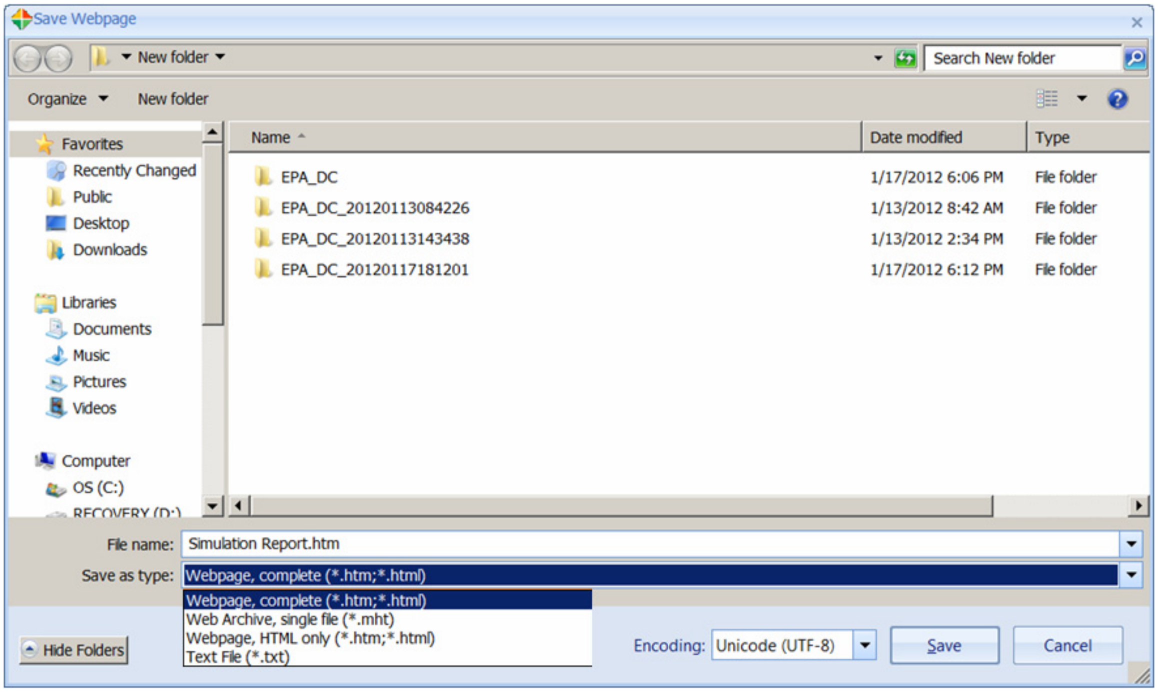
The report file is created.



3. To save the report file as *.htm, *.html, or *.mht format, select File > Save As (*.htm)...



The **Save Webpage** window opens.

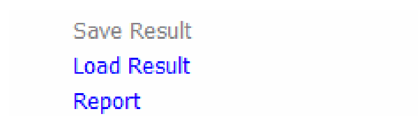


4. Select the location, file name and type.

5. Click **Save** to save the file.

Load Previously Saved Results

1. Click  to open an XtractIM workspace file *.ximx.



2. Click **Load Result** in the **Workflow** pane.

The result should be related with above *.ximx project.

3. Click **Report**.

A new report can be created.

SUMMARY OF ELECTRICAL PERFORMANCE ASSESSMENT FEATURES

<table>
<tr><th>DISTRIBUTION SYSTEM</th><th colspan="2">FEATURES</th><th>BGA PACKAGES</th><th>LEADFRAME</th></tr>
<tr><td rowspan="5">Power and Ground</td><td rowspan="2">Power-Ground Net Pair</td><td>Inductance & Capacitance</td><td>Single-die
Multi-die
Stacked-BGA</td><td>No</td></tr>
<tr><td>Broadband Impedance</td><td>Single-die
Single-BGA</td><td>No</td></tr>
<tr><td>Per-pin Property</td><td>Inductance & Resistance</td><td>Single-die
Multi-die
Stacked-BGA</td><td>No</td></tr>
<tr><td>DC Current Density</td><td>Current Density</td><td>Single-die
Single-BGA</td><td>No</td></tr>
<tr><td rowspan="3">Signal</td><td>Layout Checking</td><td>Trace / Wirebond Impedance & Coupling Coefficient</td><td>Single-die
Multi-die
Stacked BGA</td><td>Single-die
Multi-die</td></tr>
<tr><td>Net Couplings</td><td>Mutual LC and NEXT</td><td>Single-die
Single-BGA</td><td>No</td></tr>
<tr><td>Insertion and Return Loss</td><td>Insertion & Return Loss</td><td>Single-die
Single-BGA</td><td>No</td></tr>
</table>

Chapter

9

TCL Command Support for Workspace Setup

This chapter takes you through the steps to use TCL command for workspace setup in the XtractIM tool.

PREPARE FOR THE SIMULATION

Get the following issues ready before you begin the simulation.

- The Bump diameters, length, heights and conductivity
- The Stackup information
- The files to be simulated in SPD format

LESSON ONE: RUNNING TCL COMMAND FOR WORKSPACE SETUP

Introduction

TCL (Tool Command Language) scripts can be used to configure and automate frequently used command in Allegro Sigrity tools. They can also be launched outside the Sigrity tools in batch mode to automate company design flows.

New TCL command is developed to support workspace setting in XtractIM.

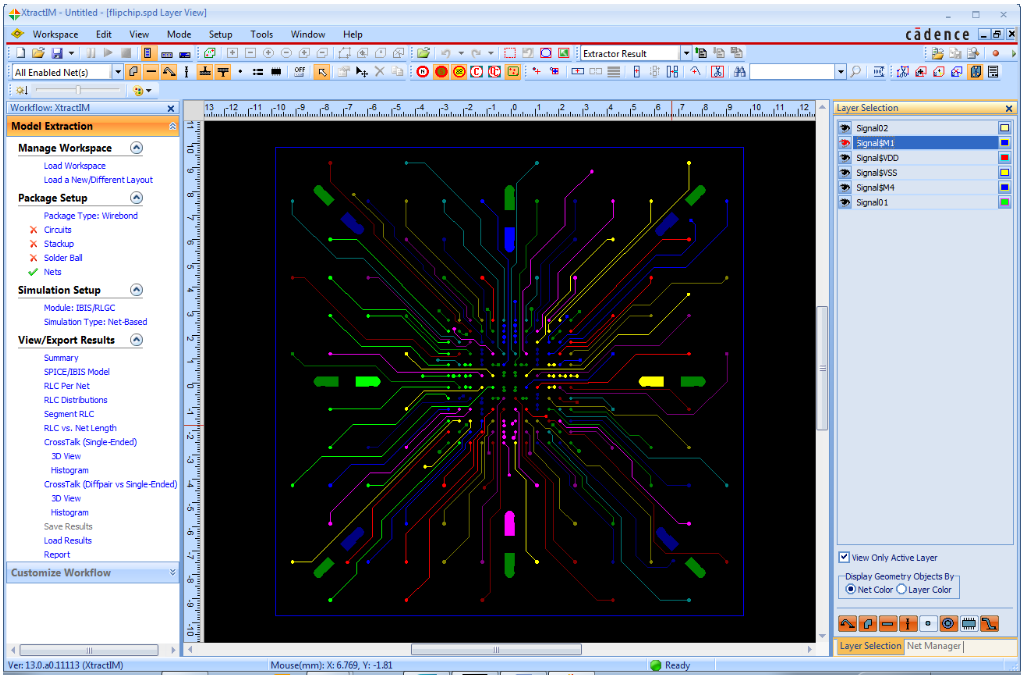
For the other existing TCL command, please refer to ***Tcl_Scripting_Reference.pdf*** in <ASI_INSTALL_DIR>\Update3\Doc\Common Documents\ for details.

Three package types are used in this tutorial to practice the new TCL command.

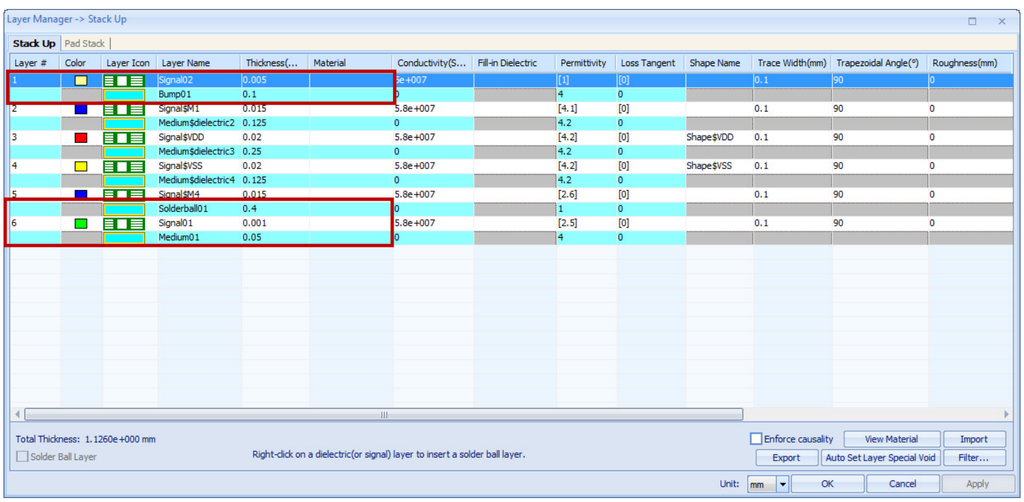
Single-Die BGA Package Sample

1. Load ***flipchip.spd***.

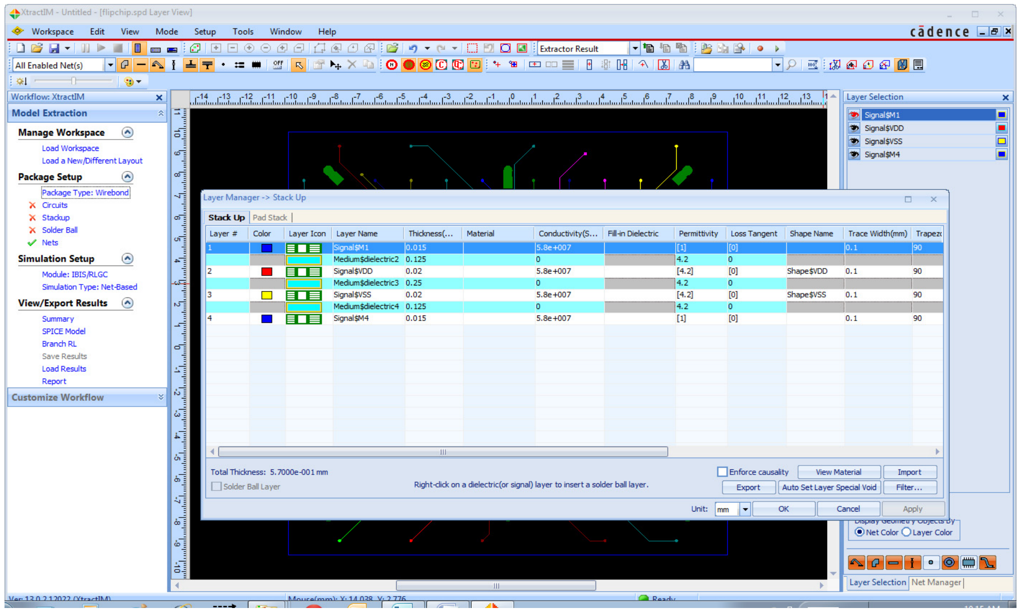
   It is typically located in <ASI_INSTALL_DIR>\Update3\SpeedXP\Samples\XtractIM\Single-Die_Sample_files\.



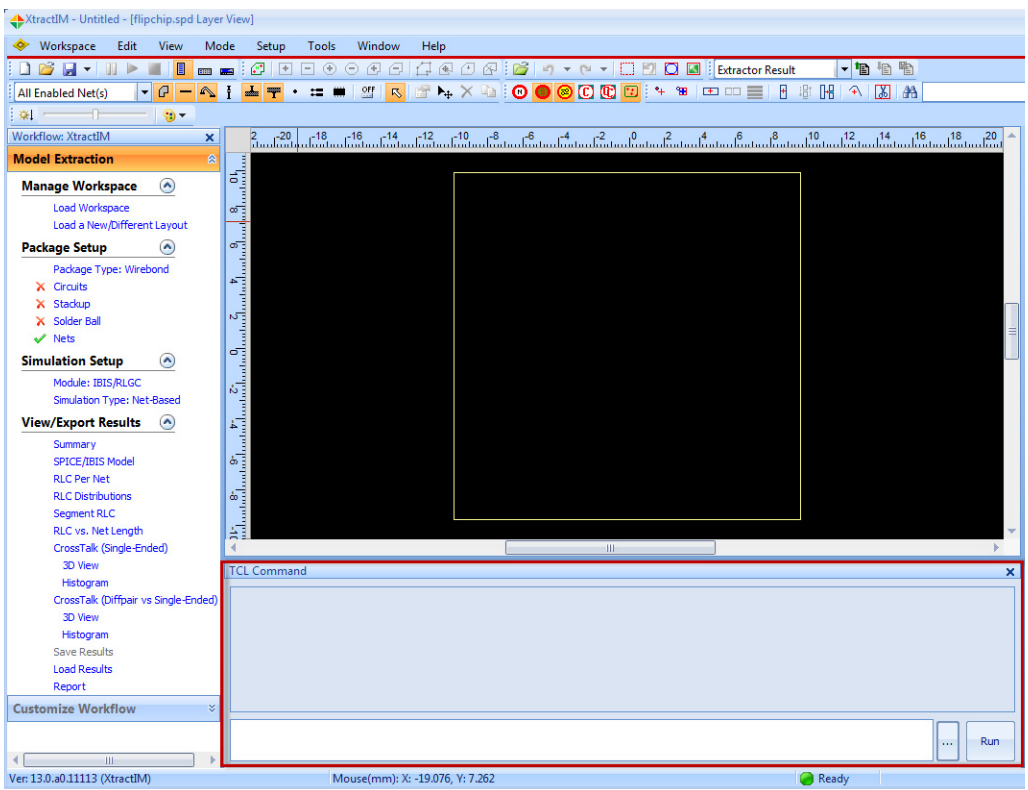
2. Remove the existing bump and solder ball layers if any before running TCL command.



A fresh example is like the following.



3. Choose View > Pane > TCL Command to open the TCL command input console.



4. Copy the following TCL command to the **TCL Command** console.

sigrity::update PackageType -dieType {0} -bgaType {0} -attachType {1}

sigrity::update Circuits -dieCkts {U1} -boardCkts {BGA1}

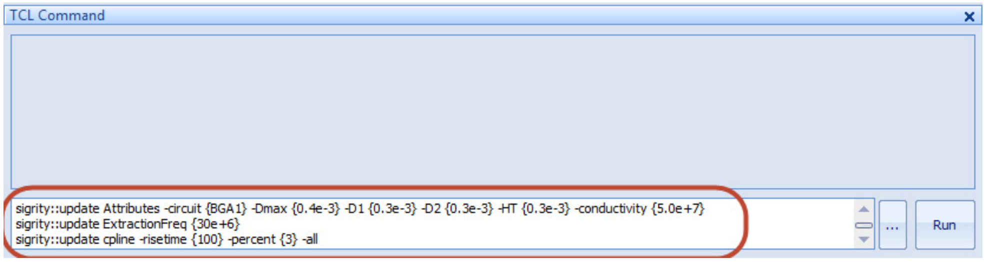
sigrity::add Layer {bump} -above {Signal$M1} -circuit {U1} sigrity::add Layer {ball} -under {Signal$M4} -circuit {BGA1}

sigrity::update Attributes -circuit {U1} -Dmax {0.1e-3} -D1 {0.08e-3} -D2 {0.08e-3} -HT {0.1e-3} -conductivity {5.0e+7}

sigrity::update Attributes -circuit {BGA1} -Dmax {0.4e-3} -D1 {0.3e-3} -D2 {0.3e-3} -HT {0.3e-3} -conductivity {5.0e+7}

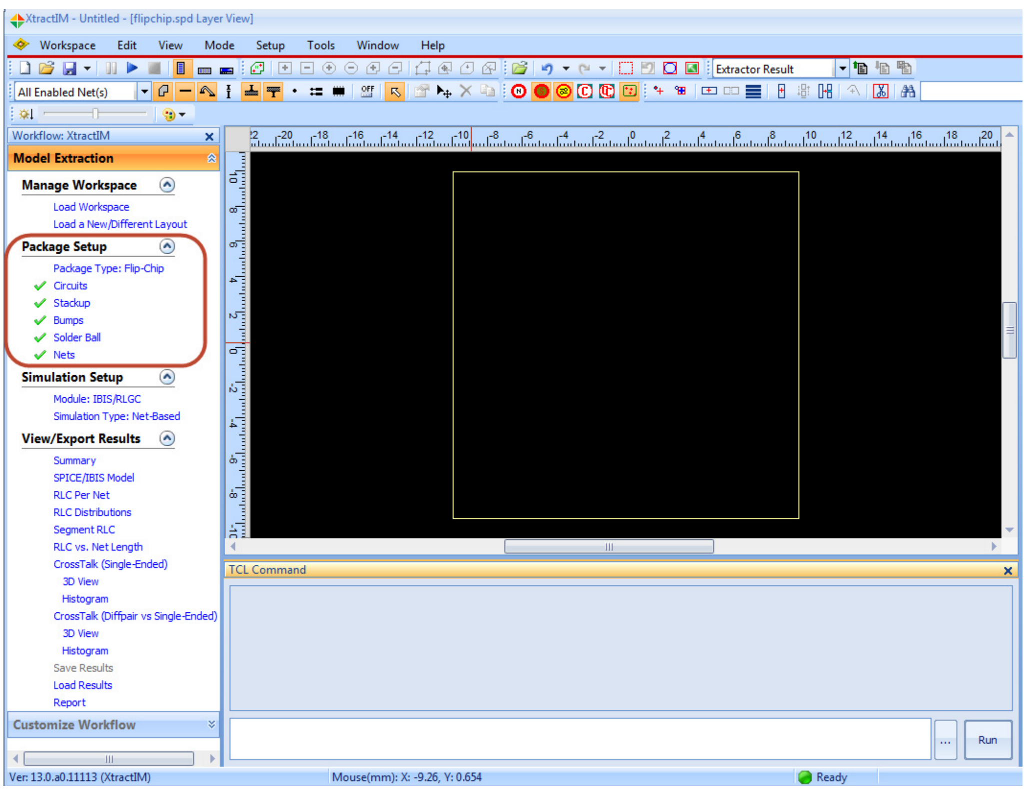
sigrity::update ExtractionFreq {30e+6}

sigrity::update cpline -risetime {100} -percent {3} –all



5. Click the ***Run*** button in the **TCL Command** console.

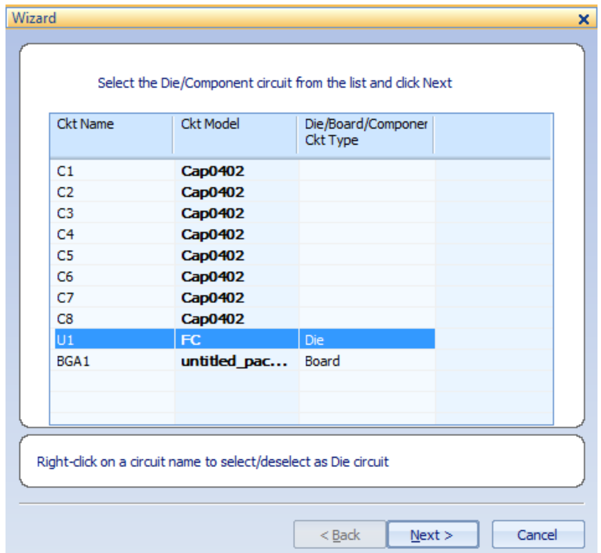
The **Package Setup** is automated.



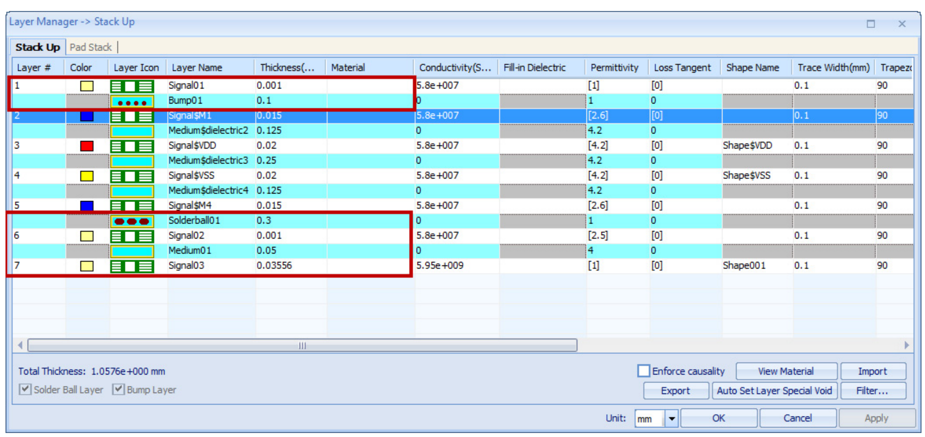
The relationship between individual TCL command and the corresponding Package Setup is described as follows.

- sigrity::update PackageType -dieType {0} -bgaType {0} -attachType {1}

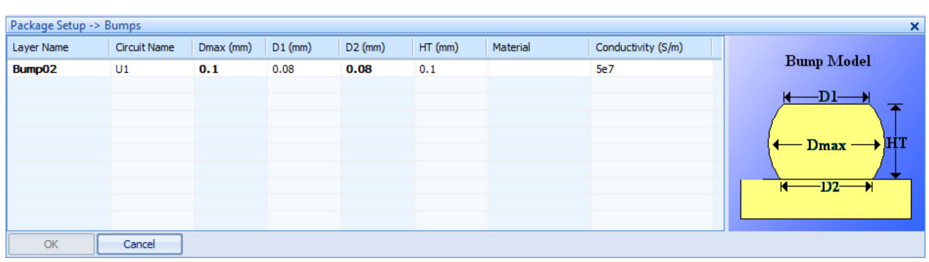
- sigrity::update Circuits -dieCkts {U1} -boardCkts {BGA1}



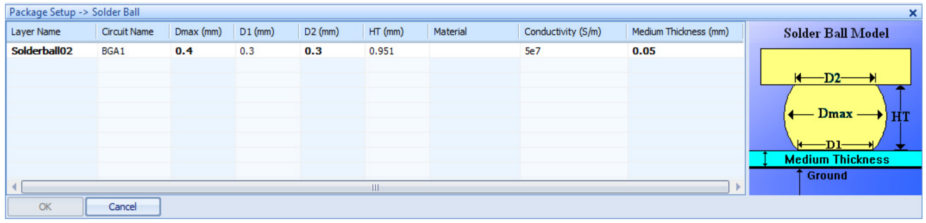
- sigrity::add Layer {bump} -above {Signal$M1} -circuit {U1}
- sigrity::add Layer {ball} -under {Signal$M4} -circuit {BGA1}



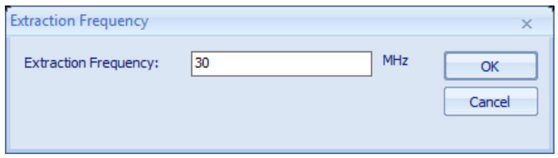
- sigrity::update Attributes -circuit {U1} -Dmax {0.1e-3} -D1 {0.08e-3} -D2 {0.08e-3} -HT {0.1e-3} -conductivity {5.0e+7}



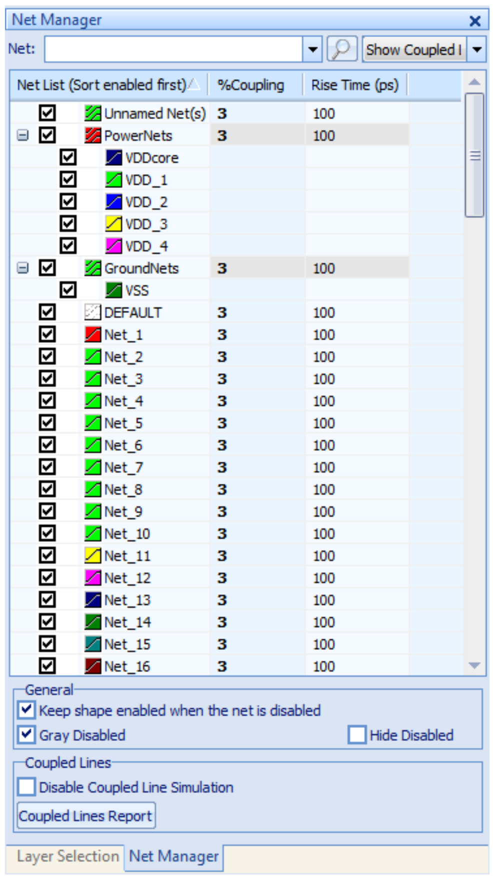
- sigrity::add Layer {ball} -under {Signal$M4} -circuit {BGA1}



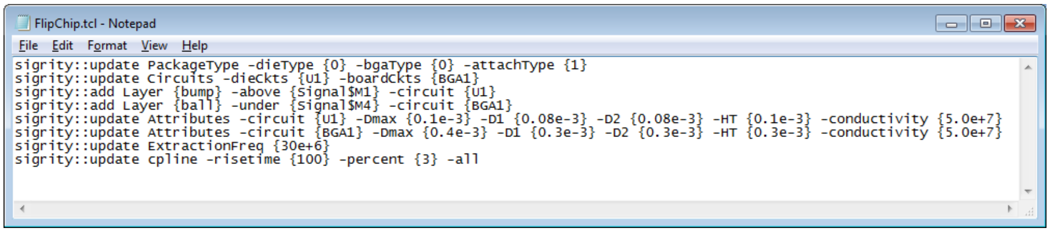
- sigrity::update ExtractionFreq {30e+6}



- sigrity::update cpline -risetime {100} -percent {3} –all

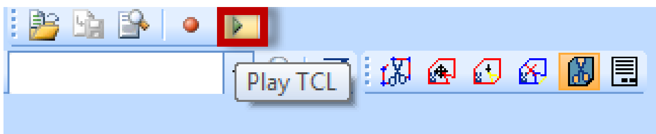


6. The TCL command used above can also be prepared in a `.tcl` file.



7. Click the **Open a TCL file** icon to load in the prepared `*.tcl` file

8. Click the **Play TCL** icon to run the prepared TCL command.



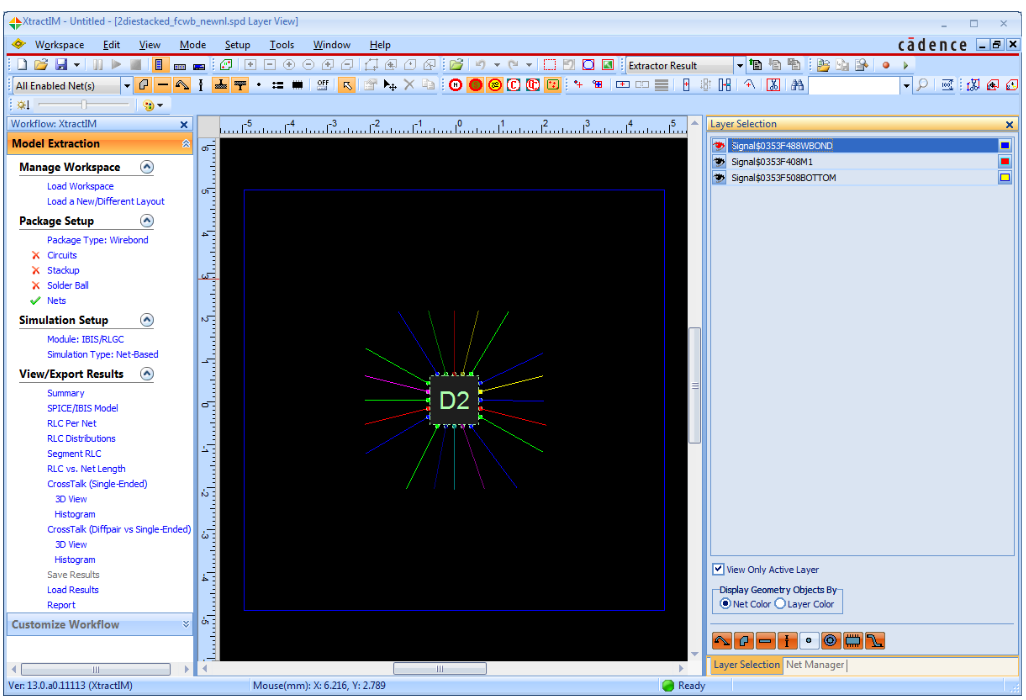
The same result will come out as in ***Step 5***.

Multiple-Die BGA Package Sample

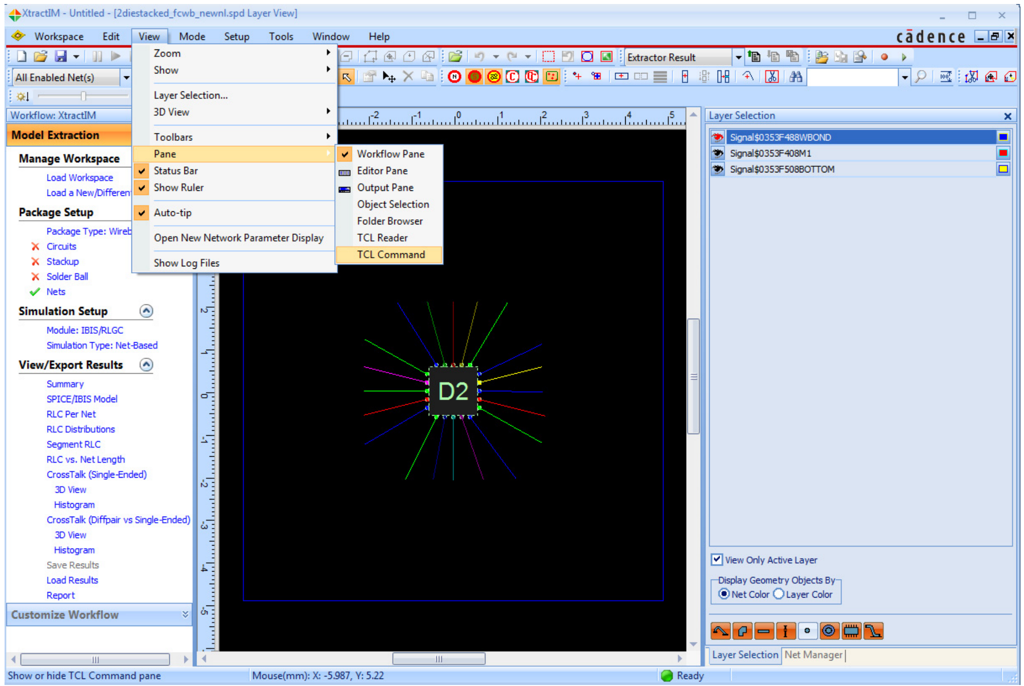
1. Load ***2diestacked_fcwb_newnl.spd***.

   It is typically located in <ASI_INSTALL_DIR>\Update3\SpeedXP\Samples\XtractIM\Single-Die_Sample_files\.

   A wirebond die is placed above a flipchip die on a BGA package.



2. Choose View > Pane > TCL Command to open the TCL command input console.



3. Copy the following TCL command to the **TCL Command** console.

sigrity::update PackageType -dieType {1} -bgaType {0} -attachType {3}

sigrity::update Circuits -dieCkts {D1} -boardCkts {BGA1} -wbCkts {D2}

sigrity::add Layer {bump} -above {Signal$0353F408M1} -circuit {D1}

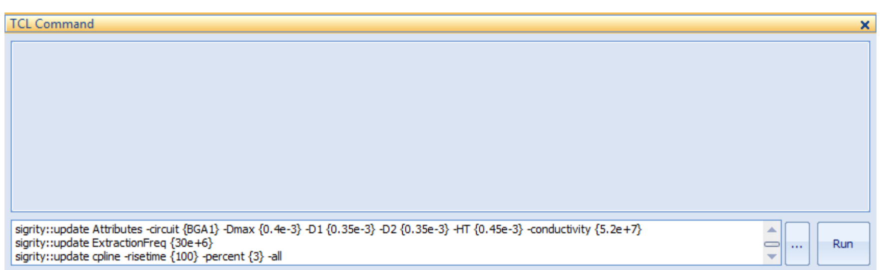
sigrity::add Layer {ball} -under {Signal$0353F508BOTTOM} -circuit {BGA1}

sigrity::update Attributes -circuit {D1} -Dmax {0.12e-3} -D1 {0.10e-3} -D2 {0.10e-3} -HT {0.08e-3} -conductivity {5.0e+7}

sigrity::update Attributes -circuit {BGA1} -Dmax {0.4e-3} -D1 {0.35e-3} -D2 {0.35e-3} -HT {0.45e-3} -conductivity {5.2e+7}

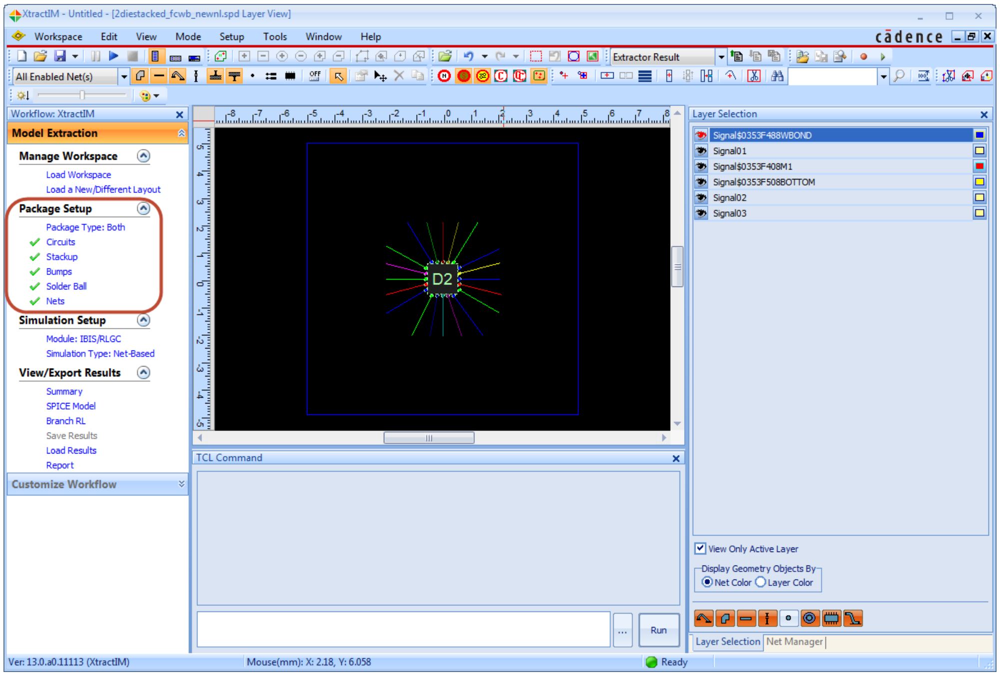
sigrity::update ExtractionFreq {30e+6}

sigrity::update cpline -risetime {100} -percent {3} –all



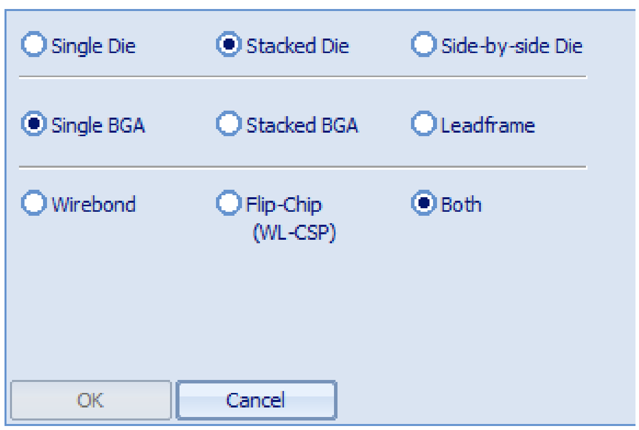
4. Click the ***Run*** button in the **TCL Command** console.

The **Package Setup** is automated.

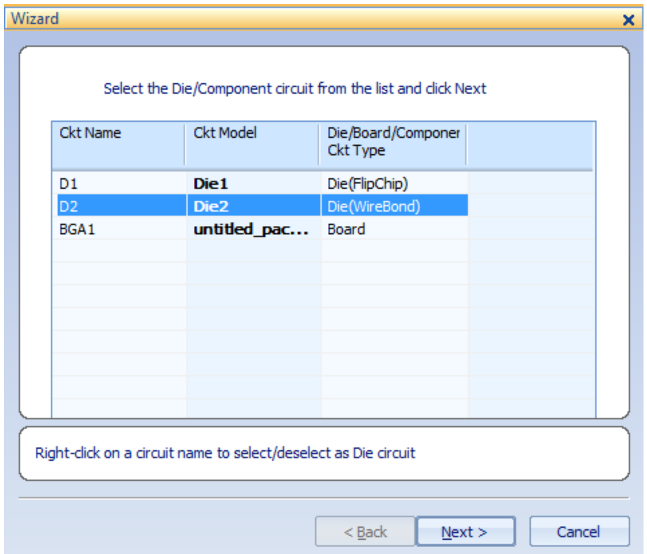


The relationship between individual TCL command and the corresponding Package Setup is described as follows.

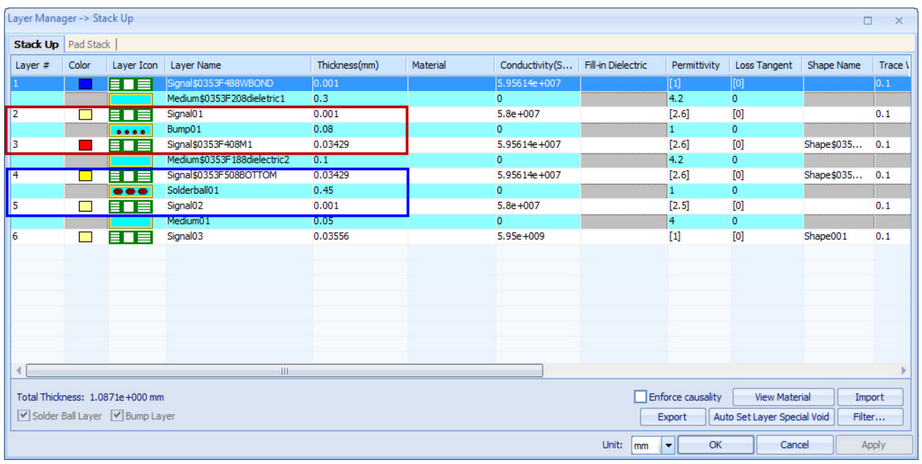
- sigrity::update PackageType -dieType {1} -bgaType {0} -attachType {3}



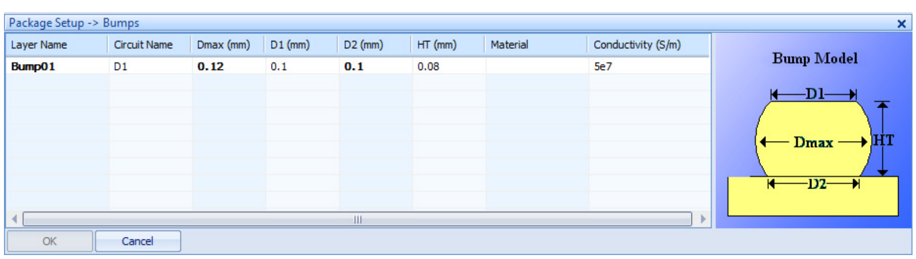
- sigrity::update Circuits -dieCkts {D1} -boardCkts {BGA1} -wbCkts {D2}



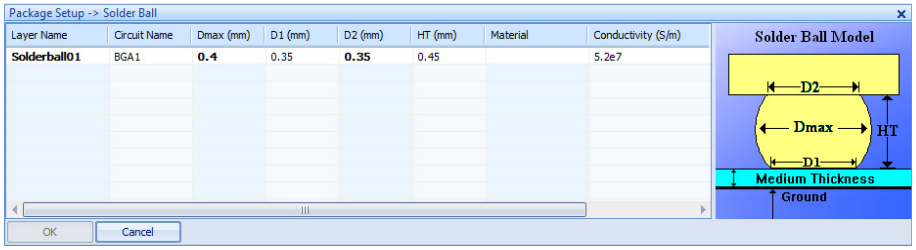
- sigrity::add Layer {bump} -above {Signal$0353F408M1} -circuit {D1}
- sigrity::add Layer {ball} -under {Signal$0353F508BOTTOM} -circuit {BGA1}



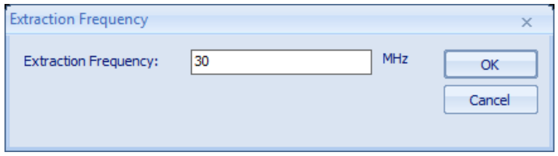
- sigrity::update Attributes -circuit {D1} -Dmax {0.12e-3} -D1 {0.10e-3} -D2 {0.10e-3} -HT {0.08e-3} -conductivity {5.0e+7}



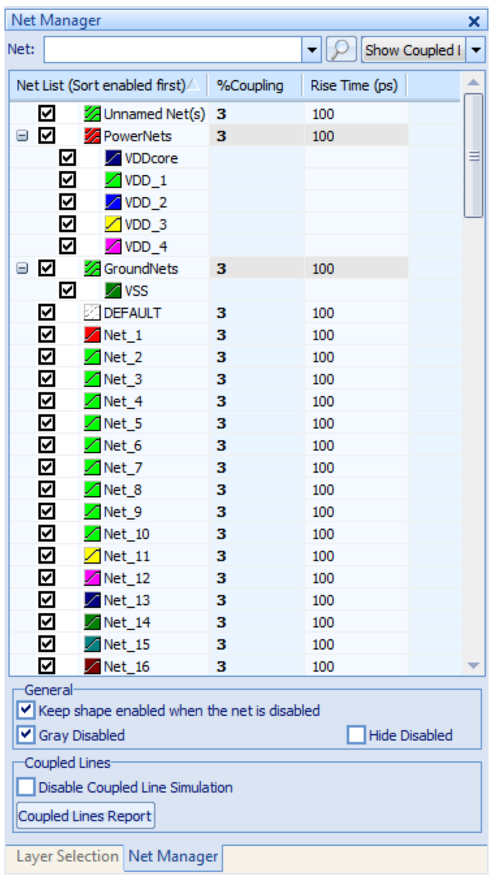
- sigrity::update Attributes -circuit {BGA1} -Dmax {0.4e-3} -D1 {0.35e-3} -D2 {0.35e-3} -HT {0.45e-3} -conductivity {5.2e+7}



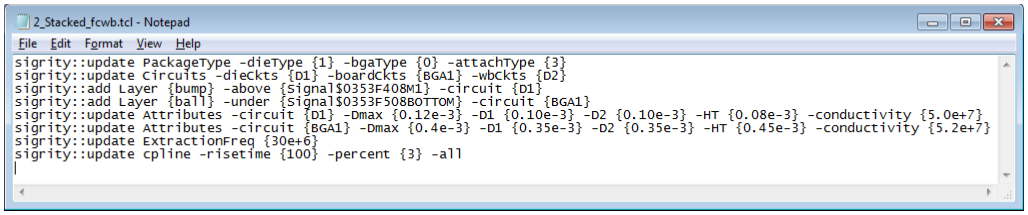
- sigrity::update ExtractionFreq {30e+6}



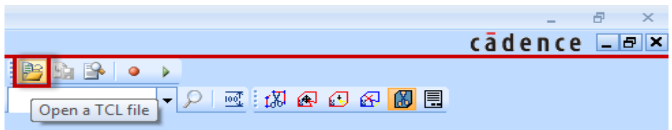
- sigrity::update cpline -risetime {100} -percent {3} –all



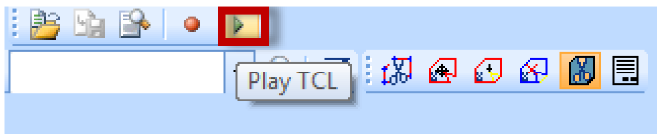
5. The TCL command used above can also be prepared in a `.tcl` file.



6. Click the **Open a TCL file** icon to load the prepared `*.tcl` file.



7. Click the **Play TCL** icon to run the prepared TCL command.



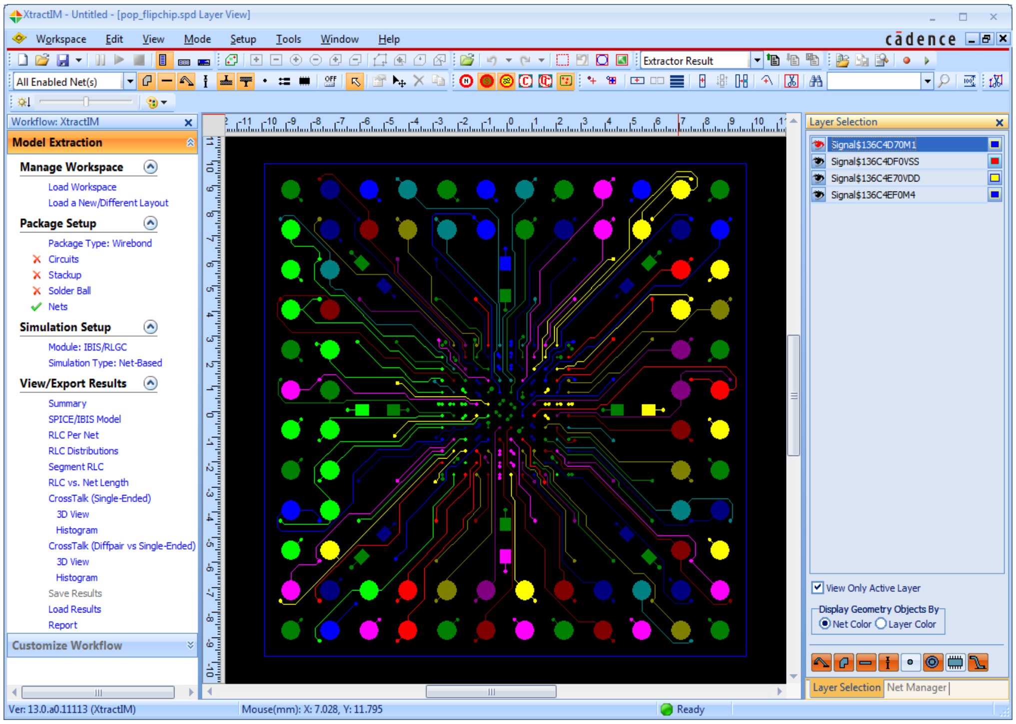
The same result will come out as in ***Step 4***.

Stacked-BGA Package Sample

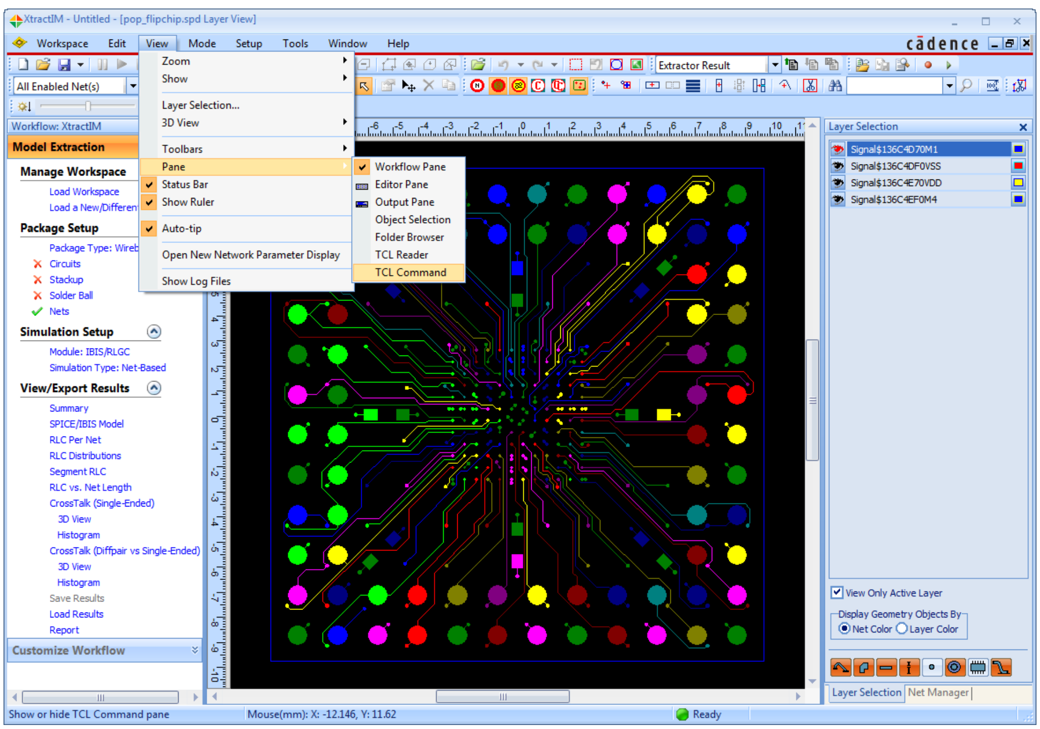
1. Load ***pop_flipchip.spd***.

   It is typically located in <ASI_INSTALL_DIR>\Update3\SpeedXP\Samples\XtractIM\Single-Die_Sample_files\.

   A flipchip BGA package is placed above a flipchip BGA package.



2. Choose View > Pane > TCL Command to open the TCL command input console.



3. Copy the following TCL command to the **TCL Command** console.

sigrity::update PackageType -dieType {0} -bgaType {1} -attachType {1}

sigrity::update Circuits -dieCkts {U1} -boardCkts {POPtop} -boardCkts {POPbottom} sigrity::add Layer {bump} -above {Signal$136C4D70M1} -circuit {U1}

sigrity::add Layer {ball} -above {Signal$136C4D70M1} -circuit {POPtop}

sigrity::add Layer {ball} -under {Signal$136C4EF0M4} -circuit {POPbottom}

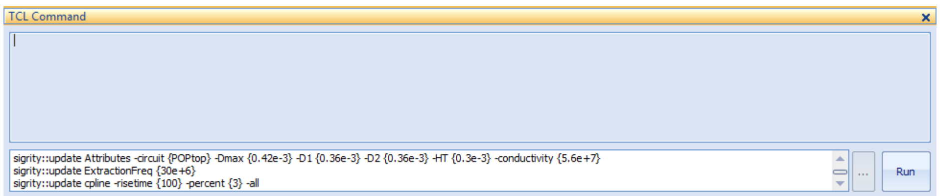
sigrity::update Attributes -circuit {U1} -Dmax {0.12e-3} -D1 {0.10e-3} -D2 {0.10e-3} -HT {0.08e-3} -conductivity {5.0e+7}

sigrity::update Attributes -circuit {POPbottom} -Dmax {0.4e-3} -D1 {0.35e-3} -D2 {0.35e-3} -HT {0.45e-3} -conductivity {5.2e+7}

sigrity::update Attributes -circuit {POPtop} -Dmax {0.42e-3} -D1 {0.36e-3} -D2 {0.36e-3} -HT {0.3e-3} -conductivity {5.6e+7}

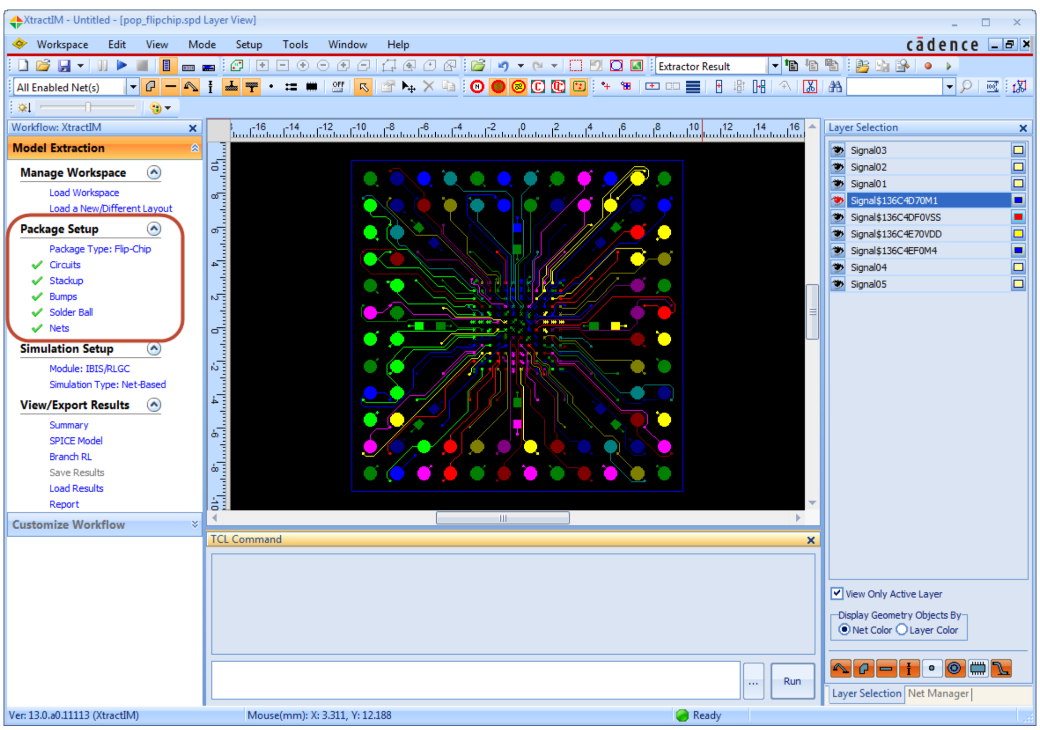
sigrity::update ExtractionFreq {30e+6}

sigrity::update cpline -risetime {100} -percent {3} -all



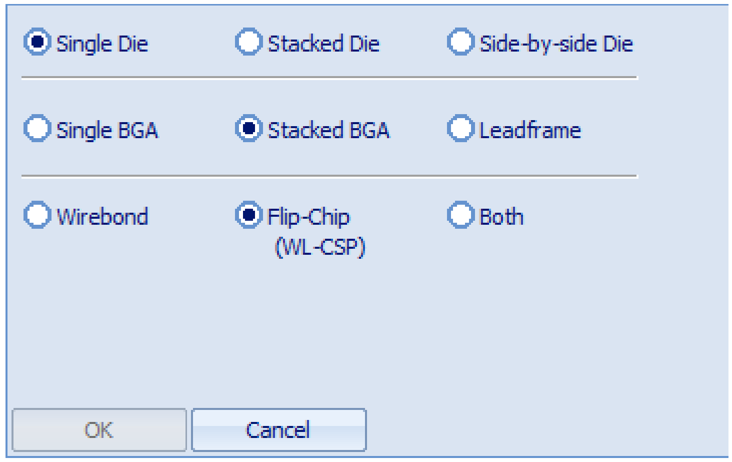
4. Click the ***Run*** button in the **TCL Command** console.

   The **Package Setup** is automated.

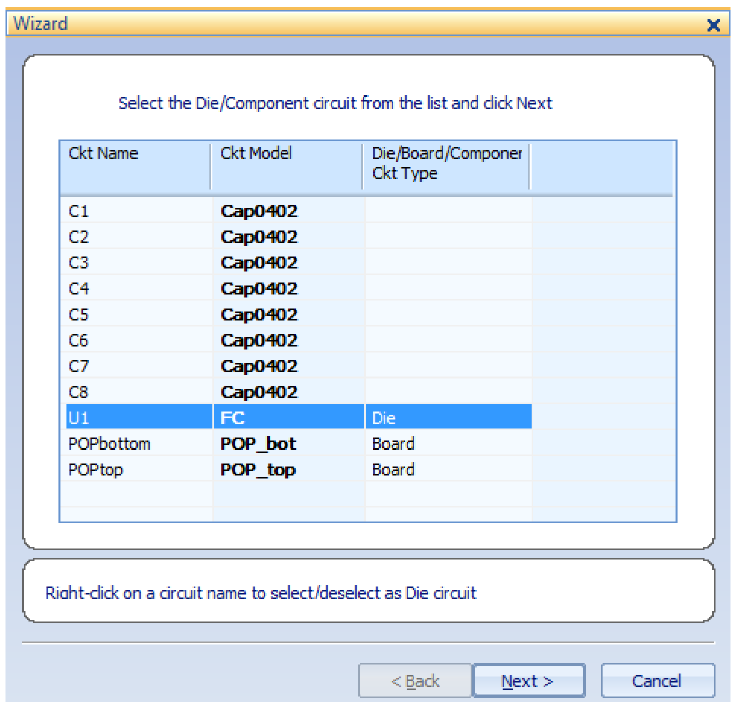


The relationship between individual TCL command and the corresponding Package Setup is described as follows.

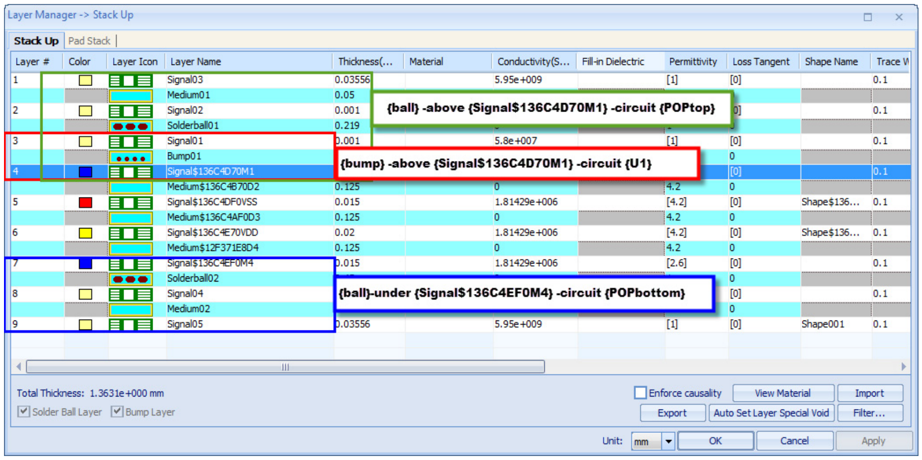
- sigrity::update PackageType -dieType {0} -bgaType {1} -attachType {1}



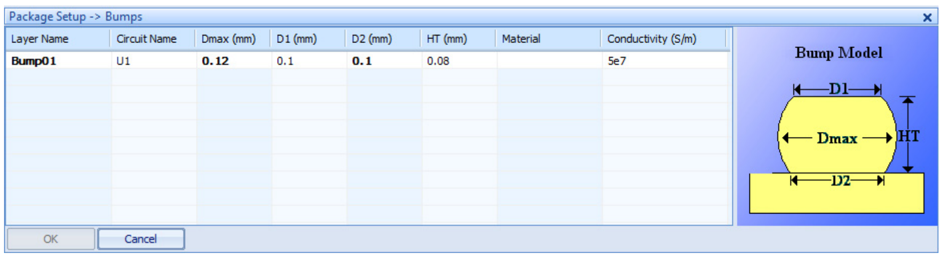
- sigrity::update Circuits -dieCkts {U1} -boardCkts {POPtop} -boardCkts {POPbottom}



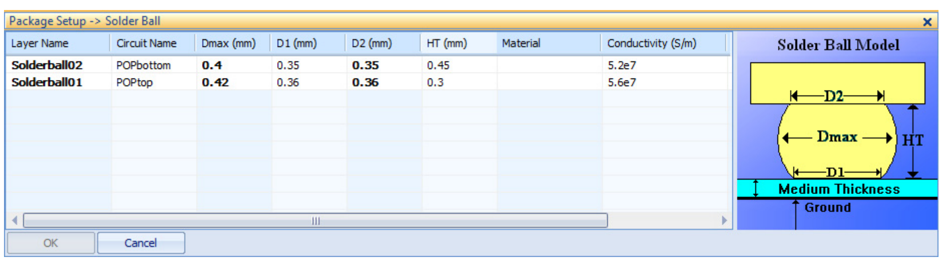
- sigrity::add Layer {bump} -above {Signal$136C4D70M1} -circuit {U1}
- sigrity::add Layer {ball} -above {Signal$136C4D70M1} -circuit {POPtop}
- sigrity::add Layer {ball} -under {Signal$136C4EF0M4} -circuit {POPbottom}



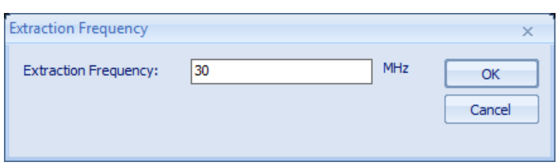
- sigrity::update Attributes -circuit {U1} -Dmax {0.12e-3} -D1 {0.10e-3} -D2 {0.10e-3} -HT {0.08e-3} -conductivity {5.0e+7}



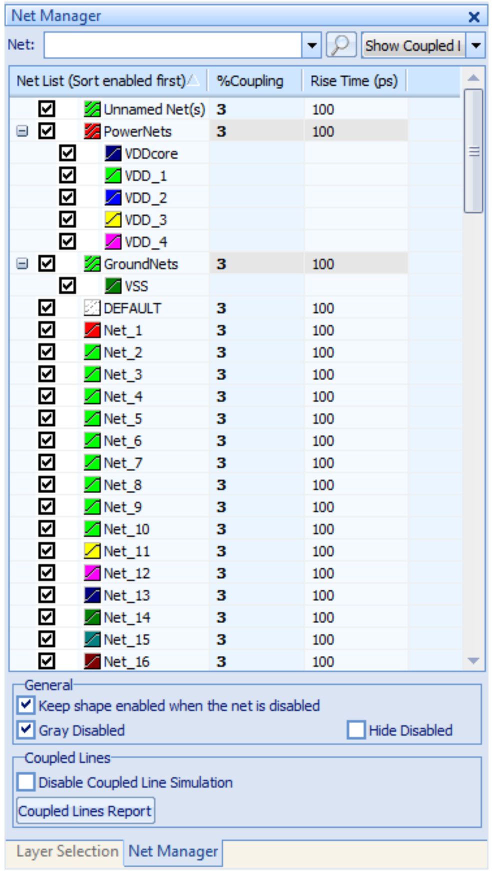
- sigrity::update Attributes -circuit {POPbottom} -Dmax {0.4e-3} -D1 {0.35e-3} -D2 {0.35e-3} -HT {0.45e-3} -conductivity {5.2e+7}
- sigrity::update Attributes -circuit {POPtop} -Dmax {0.42e-3} -D1 {0.36e-3} -D2 {0.36e-3} -HT {0.3e-3} -conductivity {5.6e+7}



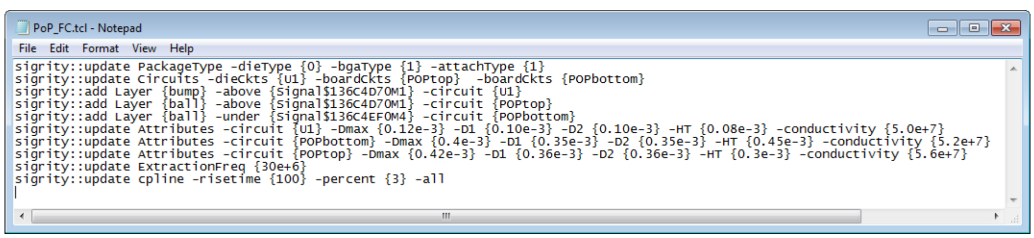
- sigrity::update ExtractionFreq {30e+6}



- sigrity::update cpline -risetime {100} -percent {3} –all



5. The TCL command used above can also be prepared as a `.tcl` file.

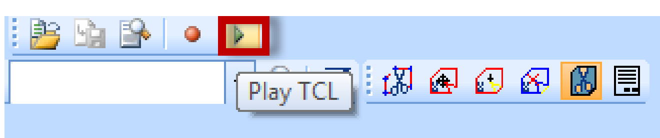

PoP_FC.tcl - Notepad

File Edit Format View Help

```
sigrity::update PackageType -dieType {0} -bgaType {1} -attachType {1}
sigrity::update Circuits -dieCkts {U1} -boardCkts {POPtop}  -boardCkts {POPbottom}
sigrity::add Layer {bump} -above {Signal$136C4D70M1} -circuit {U1}
sigrity::add Layer {ball} -above {Signal$136C4D70M1} -circuit {POPtop}
sigrity::add Layer {ball} -under {Signal$136C4EF0M4} -circuit {POPbottom}
sigrity::update Attributes -circuit {U1} -Dmax {0.12e-3} -D1 {0.10e-3} -D2 {0.10e-3} -HT {0.08e-3} -conductivity {5.0e+7}
sigrity::update Attributes -circuit {POPbottom} -Dmax {0.4e-3} -D1 {0.35e-3} -D2 {0.35e-3} -HT {0.45e-3} -conductivity {5.2e+7}
sigrity::update Attributes -circuit {POPtop} -Dmax {0.42e-3} -D1 {0.36e-3} -D2 {0.36e-3} -HT {0.3e-3} -conductivity {5.6e+7}
sigrity::update ExtractionFreq {30e+6}
sigrity::update cpline -risetime {100} -percent {3} -all
```

6. Click the **Open a TCL file** icon to load in the prepared `*.tcl` file.

7. Click the **Play TCL** icon to run the prepared TCL command.



This same result will come out as in ***Step 4***.

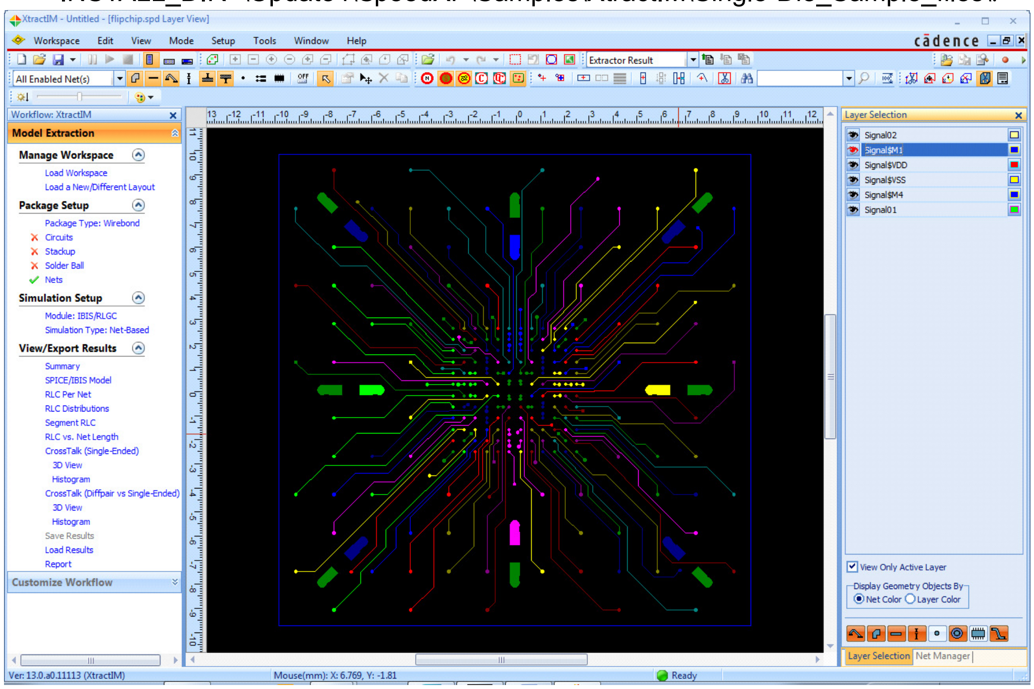
LESSON TWO: TCL COMMAND RECORDING SUPPORT FOR WORKSPACE SETUP

In this lesson, the TCL recording function for XtractIM workflow setup is introduced for preparing a reusable TCL file.

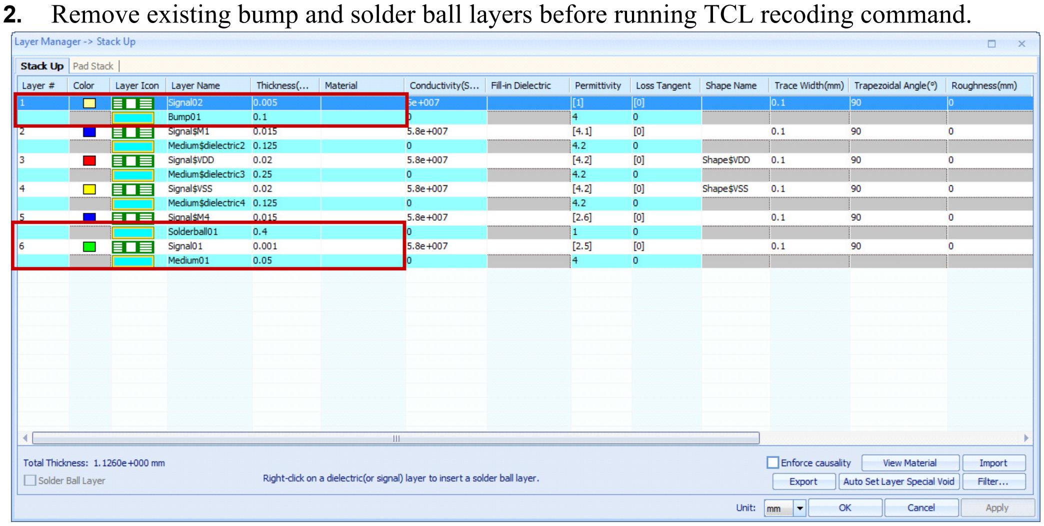
Load Design

1. Load a sample spd file.

   flipchip.spd is used as sample case in this tutorial. It is by default located at <INSTALL_DIR>\Update4\SpeedXP\Samples\XtractIM\Single-Die_Sample_files\.

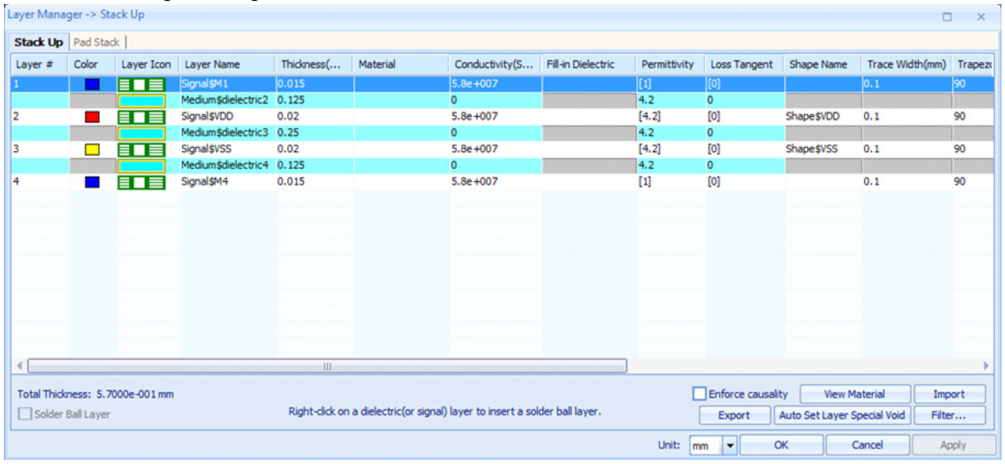


2. Remove existing bump and solder ball layers before running TCL recoding command.

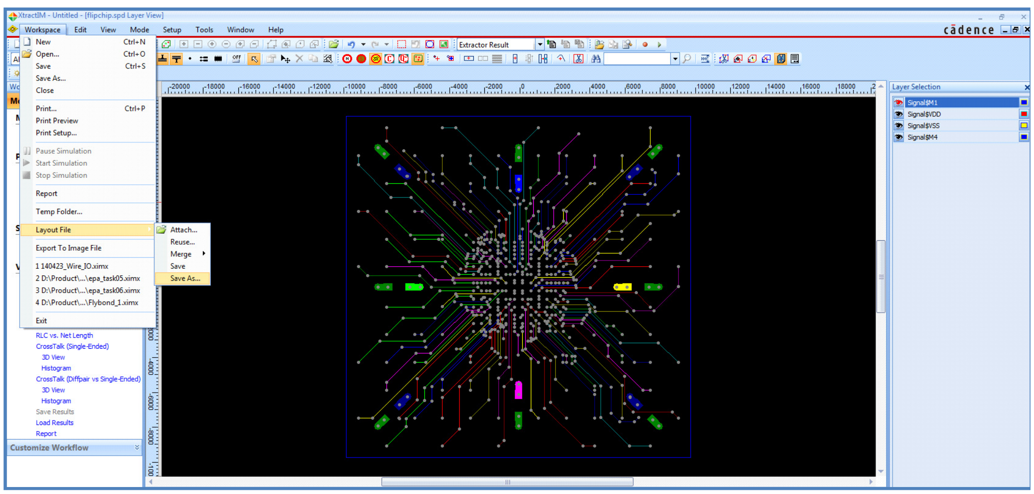


3. Click **OK**.

The Stackup is setup as below.

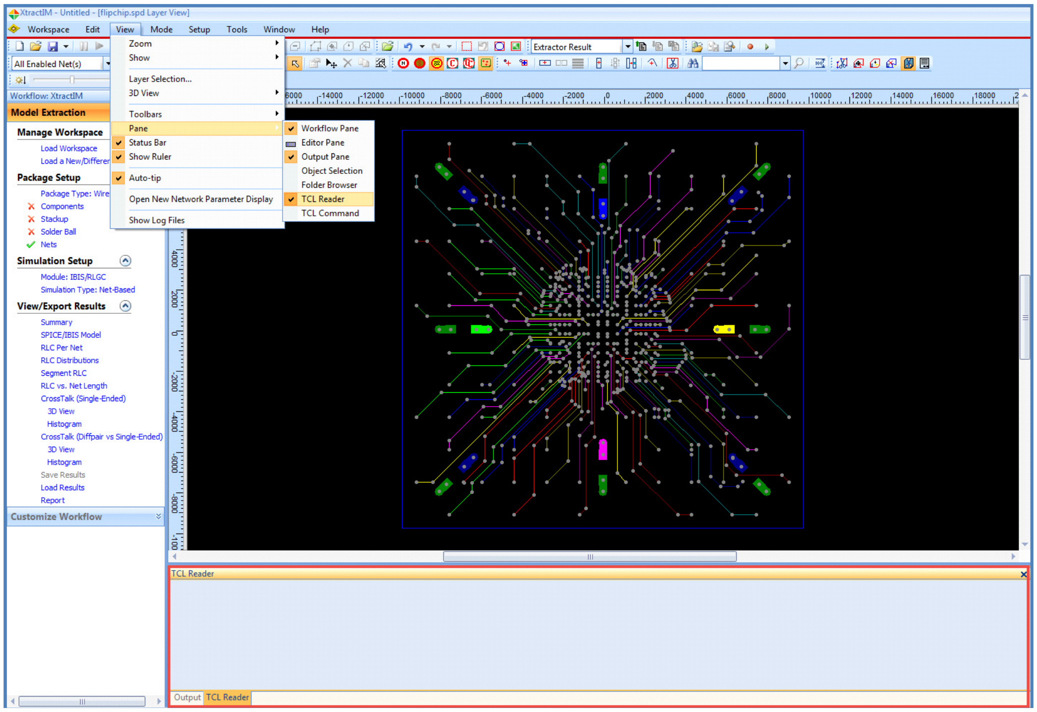


4. Save the clean spd file to the working directory.

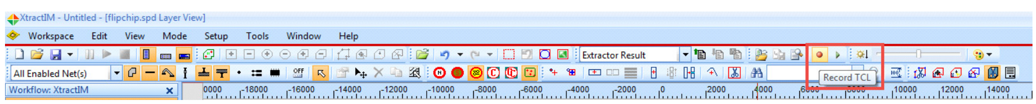


Record TCL Command for Workspace Setup

1. Choose View > Pane > TCL Reader.

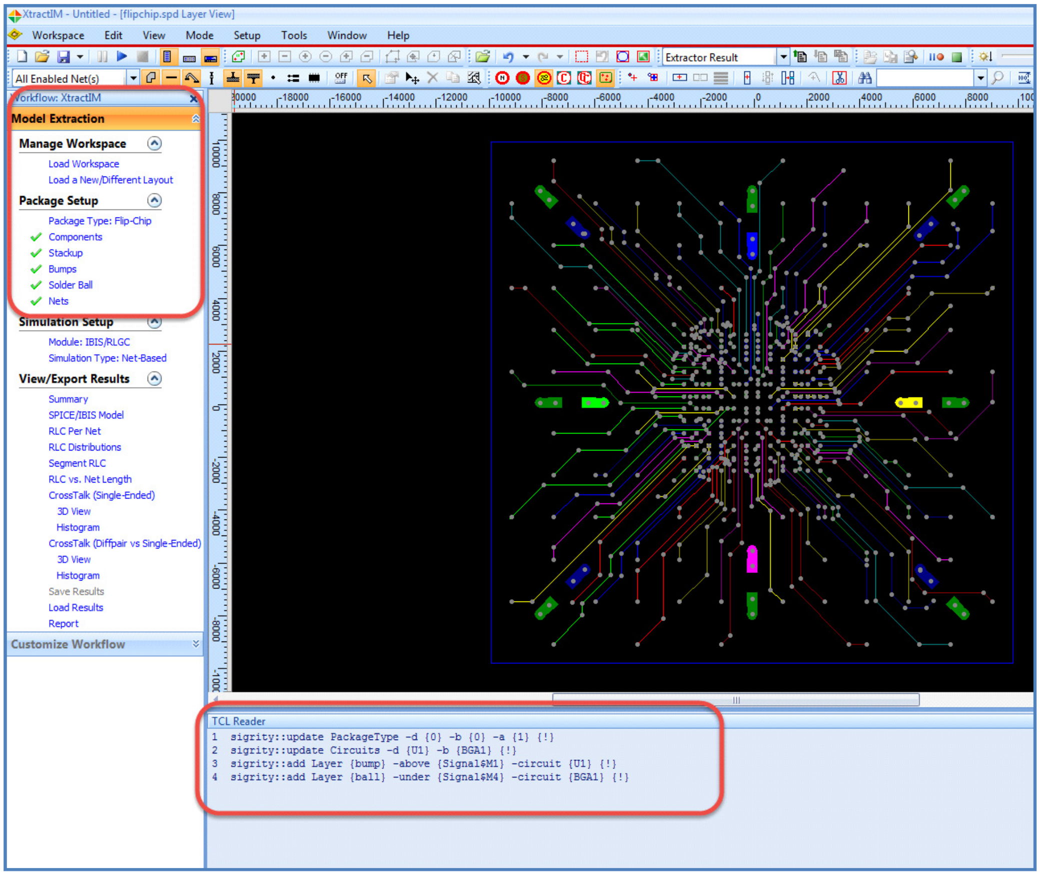


2. Click the **Record TCL** button.

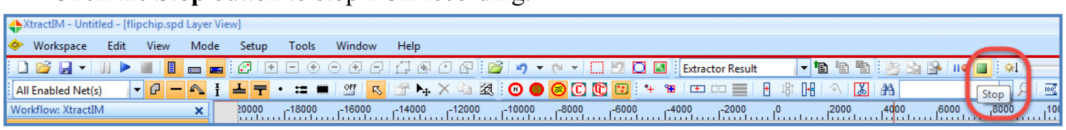


3. Set up the workflow as usual.

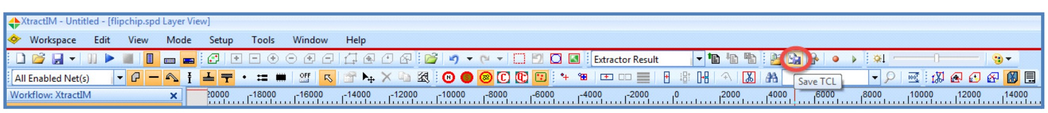
The used TCL command will be written in the TCL reader.

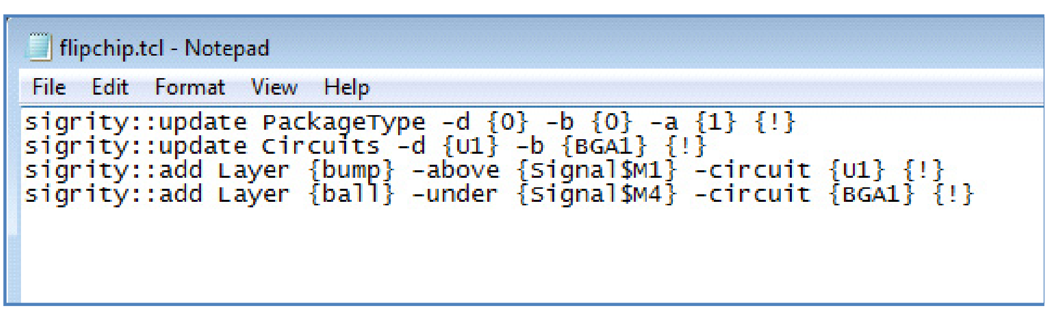


4. Click the **Stop** button to stop TCL recording.



5. Click the **Save TCL** button to save the TCL command to a file for reuse.





Chapter 10

Network Parameter Viewer

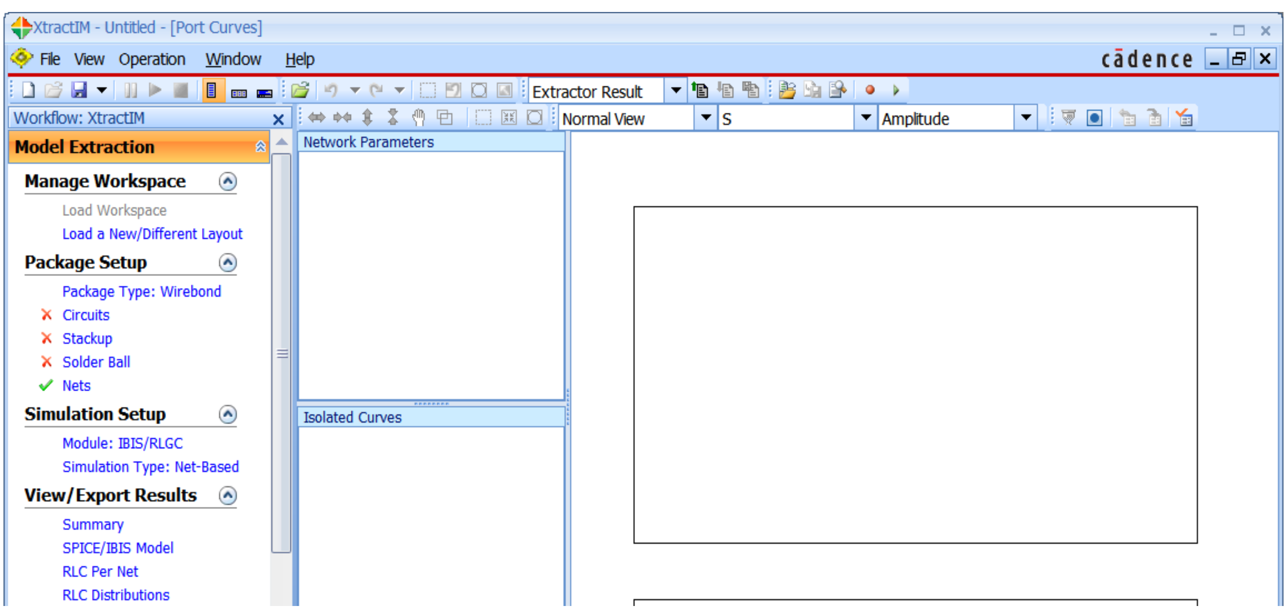
This chapter takes you through the steps to use the XtractIM tool to view Network Parameter with both BNP and Touchstone files.

Load BNP / Touchstone Files

1. Launch XtractIM.
2. Click the **New** button on the toolbar.
3. To open new Network Parameter Display, choose

   View > Open New Network Parameter Display.

   The **Network Parameters** pane appears.



4. Choose

   File > Open

   or right-click in the upper left pane.
5. Select **Load** from the Context menu.
6. Select the network parameter files to be loaded.

   Multiple BNP/ Touchstone files can be opened at the same time. Once a file is loaded successfully, there is a new entry of the Network Parameter Matrix added to the Matrix pane.
7. Click on the **Open** button.

By default the diagonal elements of the matrix are shown. The corresponding frequency curves are displayed in the Curve pane. A unique ID is assigned to the matrix and its elements, followed by a customizable display names.

The icon  denotes a matrix corresponding to a Touchstone file while the icon  denotes a matrix corresponding to a BNP file.

Select Matrix Elements

1. Select one or more parameter matrices.
2. Select

   View > Select Matrix Elements

   or right click on a selected matrix.
3. From the context menu select

   Select Matrix Elements

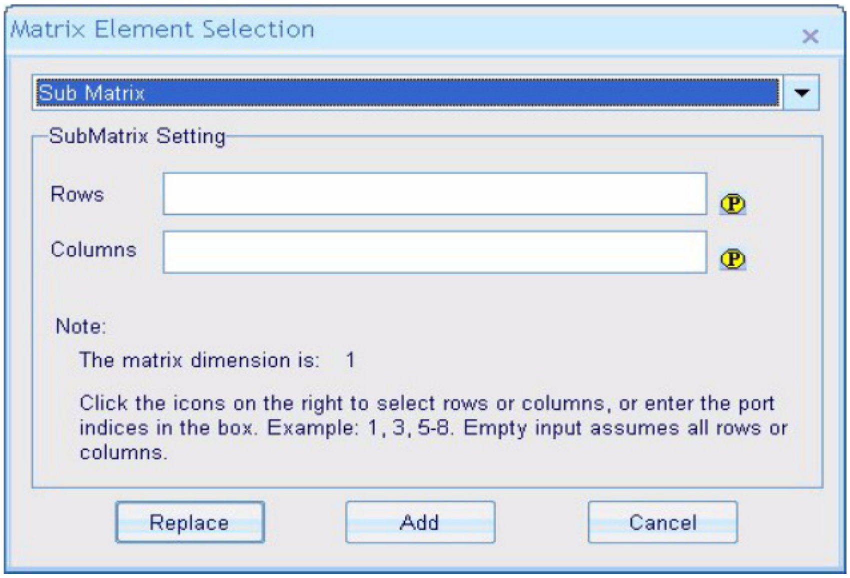
   The **Matrix Element Selection** dialog appears.
4. From the drop-down box choose from the pre-defined patterns.

   - Diagonal
   - Full Matrix
   - Lower Triangular Matrix
   - Sub Matrix
   - Upper Triangular Matrix

   The **Sub Matrix** option is not available if more than one matrix is selected and the dimensions are different
5. If **Sub Matrix** is selected, enter the row and column numbers in the input boxes;

   or click on the  icon and choose from the list of ports.
6. Click the **Replace** button to replace the existing displayed curves with the selections;

   or click the **Add** button to add the selected curves to the existing list.



Hide / Show Matrix Elements

1. Single-click on the color icon in front of a matrix element to trigger the visibility; or select multiple matrices or matrix elements.
2. Select

   View > Show/Hide Curves

   or right-click on a selected matrix or curve.
3. Select

   Show/Hide Curves
4. Choose from the available Visibility Control options.

Delete Matrix Elements

1. Select one or more matrix elements.
2. Select

   View > Delete Selected Curves

   or right-click on the selection.
3. Select

   Delete Selected Curves

   or press the **Delete** key.

Unload Networks

1. Select one or more matrices.
2. Select

File > Unload Network

or right-click on the selection.

3. To remove the selection from the Matrix pane select

   Unload Network

4. To remove all the loaded networks select

   Unload All Networks

   The simulation result cannot be unloaded.

SAVE BNP / TOUCHSTONE FILES

Save a Modified Network

A network can be modified by editing the comments of the file, or by performing matrix operations such as re-normalization or frequency truncation, etc. When a network is modified, a star is added to the icon before the matrix entry. To save the modification follow these steps.

1. Select a matrix entry that has been modified.
2. Select

   File > Save

   or right-click on the selected matrix.
3. Select **Save** from the context menu.

The modified information is saved into the original file, over-writing its contents with the exact same settings of the file.

Save Network as Another File

1. Select a matrix.
2. Select

   File > Save As

   or right-click on the selection.
3. Select

   Save As

In the **Save Curves** dialog, follow the same steps as in saving the simulation result

This feature allows you to convert between the BNP and Touchstone formats. Some of the information in the BNP file (such as AFS compression) is not supported by the Touchstone format.

If you save a BNP file with AFS compression into a Touchstone file, the matrices at discrete frequency samples is saved.

Export the Curves to Other Format

1. To save the screen shot of the curve window into a BMP file or PNG file select

File > Export to Image File

2. Select a matrix.
3. To save the displayed curves into a Excel .csv file select

   File > Export To Excel

View Network Properties

1. Select a matrix entry.
2. Select

   View > Property

   or right-click on the selection.
3. Select **Properties** from the context menu.

The **Network Property** dialog allows you to view various information of the network. You can edit the comments in the dialog.

- Data Types
- Design Information
- Differential Pair Setup
- Port Setup
- Sampling Frequencies
- User Comments

CUSTOMIZE THE NETWORK AND ELEMENT DISPLAY NAME

1. Select one or more matrix entries.
2. Select

   View > Name Display > Customize

   or right-click on the selection.
3. From the context menu select

   Name Display > Customize
4. Check the appropriate fields you want to include in the network name, matrix element name and curve legend.

   Examples of the element name and the curve legend are displayed as you change the settings.
5. Click **OK** to apply the setting to the selected matrices;

   or, to apply the setting to all matrices, select

   Save as Default

   The default settings are saved into the registry and used for the networks loaded in the future.
6. In additional to the default names, the display names can also be modified. Select an entry.
7. Select

   View > Name Display > Rename

or right-click on the selection.

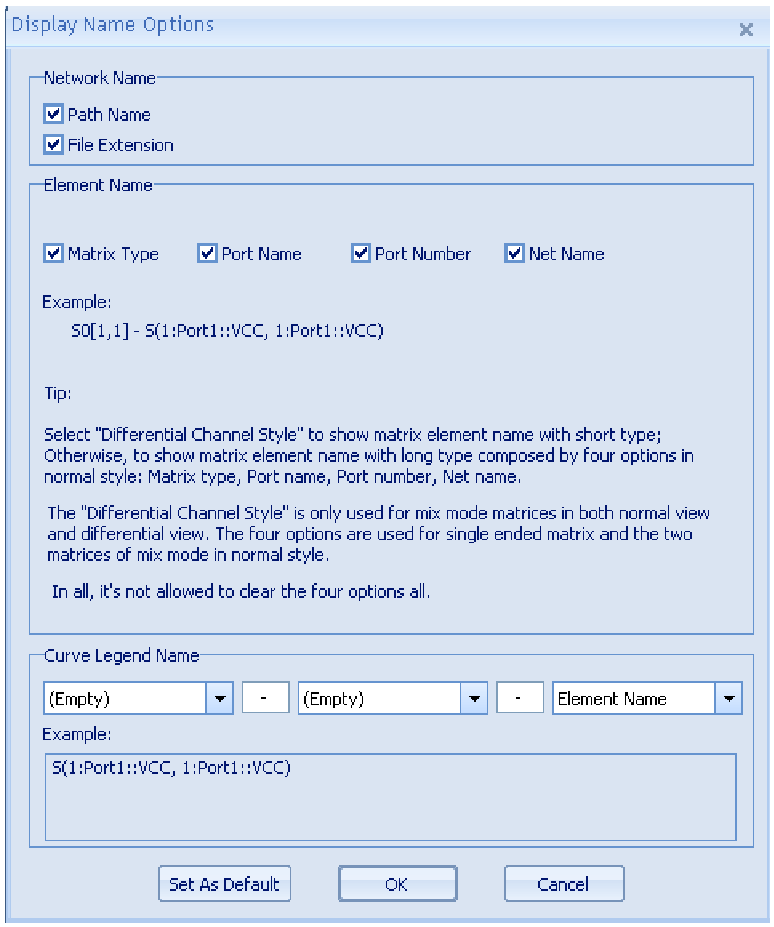
8. From the context menu select

Name Display > Rename

You can then edit the name to any string. You can switch back to the default name.

9. From the View or Context menu, select

Name Display > Default Name



Advanced Matrix Operations

Copy Network

1. Select one or more matrices.
2. Select

Operation > Copy

or right-click on the selection.

3. Select **Copy** from the context menu.When a network is copied, a new network is created. A matrix entry is added to the matrix pane.

   The status of this network is set to **modified**. It is not saved to the hard drive until you manually save it.

Network Re-normalization

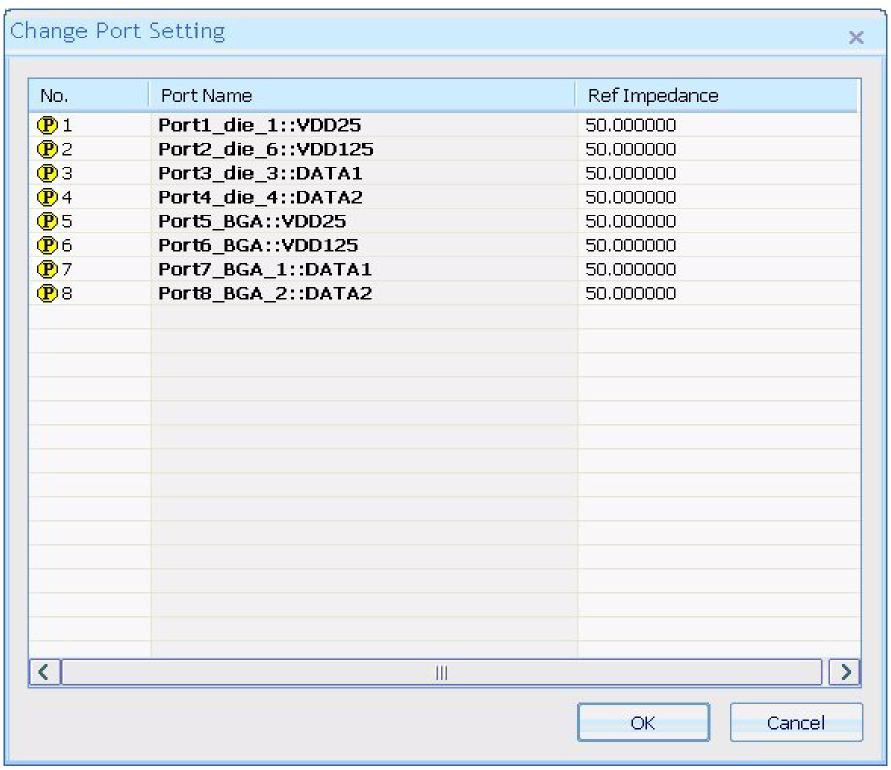
1. Select a network in the Matrix pane.
2. Select

   Operation > Re-normalize

   or right-click on the selection.
3. From the Context menu select

   Matrix Operation > Re-normalize

   The **Change Port Setting** dialog appears.
4. To edit the reference impedance of a single port select the line and click on the impedance value.
5. To change several ports at the same time, select multiple ports. A reference impedance input box appears.
6. Click **OK** to apply the changes. The **S** parameter curves are updated. The matrix is marked as modified.



Port Reduction

1. Select a network in the matrix pane.
2. Select

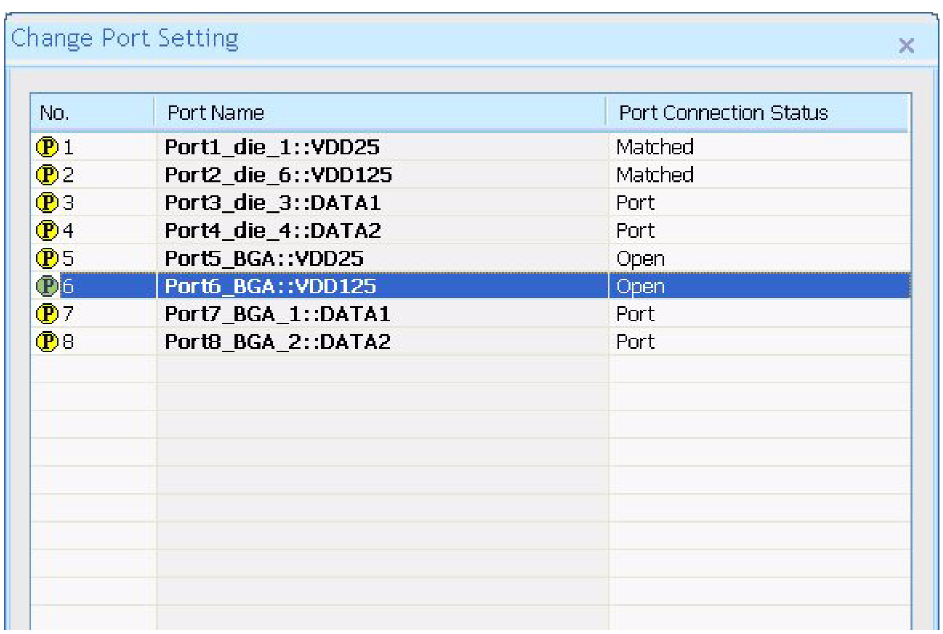
   Operation > Reduction

   or right-click on the selection.
3. From the context menu select

   Matrix Operations > Reduction

   The **Change Port Setting** dialog appears.
4. To change the status of a single port by select the line. Each port has 4 connection settings:
   - Match
   - Open
   - Remain as a Port
   - Short
5. Click on its connection status or select multiple ports.
6. Change their status together in the combo box at the bottom of the window.
7. Click **OK** to perform port reduction. The result is a new network added to the matrix pane. The network parameters are not saved to the hard drive until you manually perform a **Save**.

NOTE!	A BNP file with AFS compression cannot be reduced directly, since the AFS compression information in the original matrix is not necessarily valid for the reduced matrix. A warning message is shown. The matrix is converted into a discretely sampled format.



Passivity Enforcement

1. Select a network in the Matrix pane.
2. Select

   Operation > Passivity Enforce

   or right-click on the selection.
3. From the Context menu select

   Matrix Operations > Passivity Enforce

   A new network with passivity enforcement at each sampling frequency is generated.

NOTE!	Passivity enforcement is not available for BNP files with AFS compression.

Edit the Differential Pairs

A differential pair is defined with two nets. Pairs of differential ports can be defined under each differential pair. The differential pair and differential port information can be defined in PowerSI before the simulation and can be modified in the network parameter display.

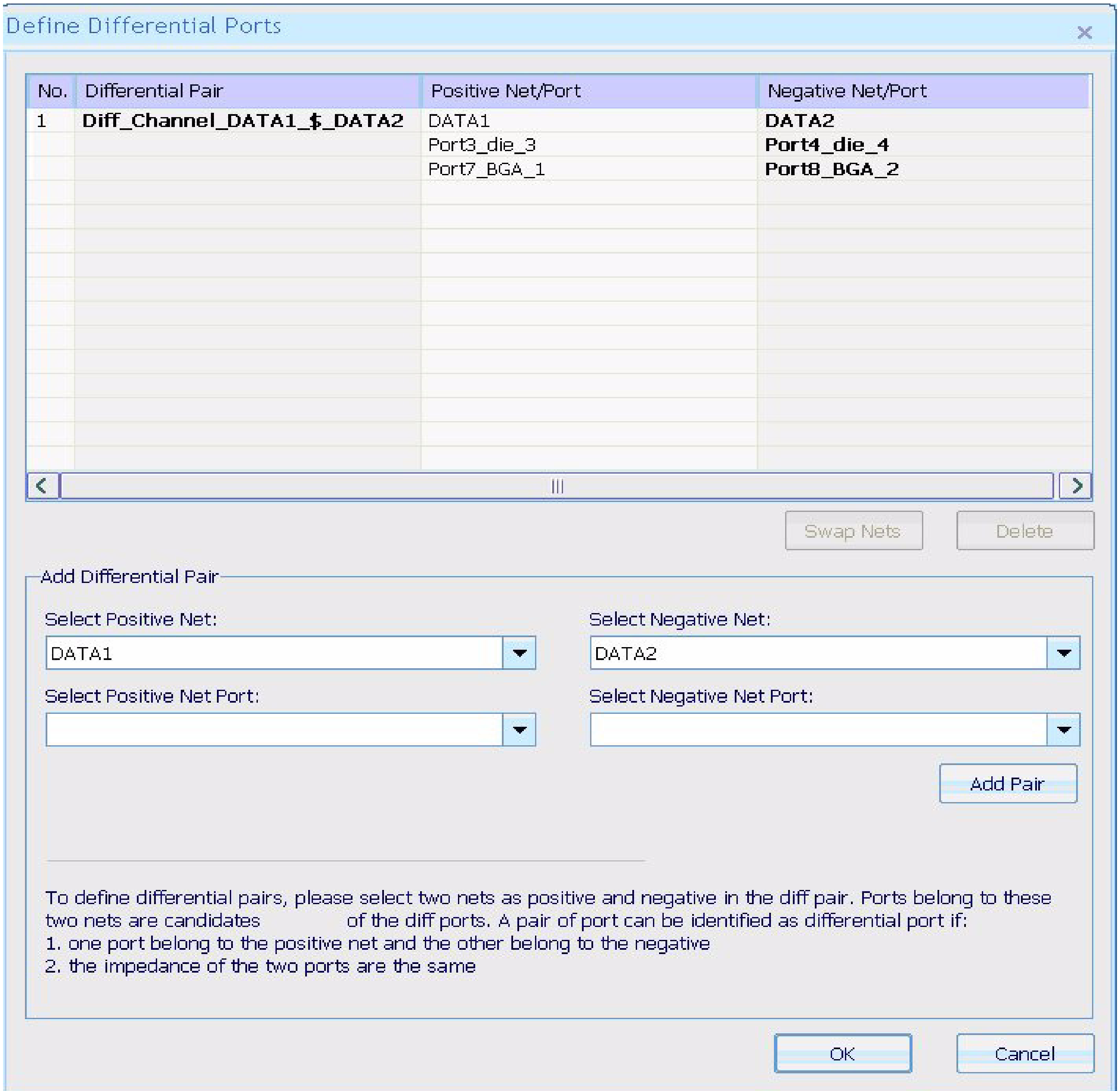
1. Select a network in the matrix pane.
2. Select

   Operation > Redefine Differential Port

   or right-click on the selection.
3. From the context menu select

   Matrix Operations > Redefine Differential Port

   The **Define Differential Ports** dialog appears.
4. To remove an existing definition, select a pair of nets or a pair of ports.
5. Click the **Delete** button.
6. To add a new pair, select a pair of nets from the net combo boxes.
7. Select a pair of ports from the port combo boxes.
8. Click **Add Pair** to create the differential ports.
9. Click **OK** to apply the changes. The mixed-mode network parameters are updated.



Redefine Mixed Mode Port

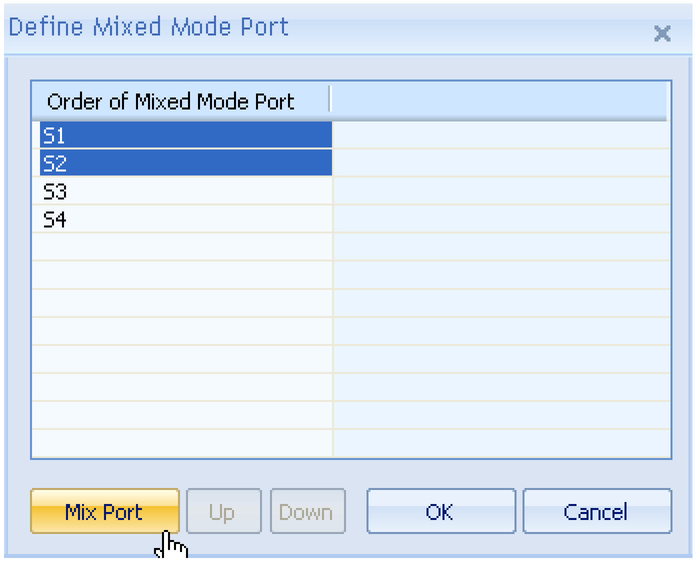
1. Select a network in the matrix pane.
2. Select

   Operation > Redefine Mixed Mode Port

   or right-click on the selection.
3. From the context menu select

   Matrix Operations > Redefine Mixed Mode Port

The **Define Mixed Mode Port** dialog appears.



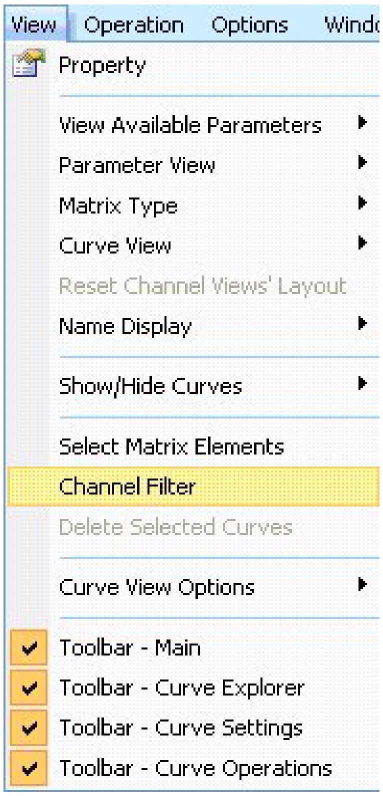
4. Hold down CTRL key and select two ports.
5. Click **Mix Port** to generate new mixed S-parameter.
6. Click **Ok** to apply changes.

   The mixed-mode network parameters are updated.

Show Network Parameters Via Channel Filter

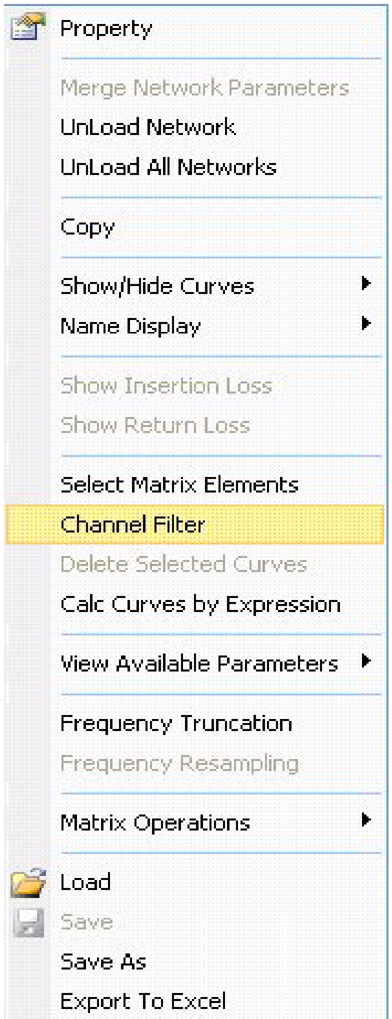
1. Select a matrix.
2. Click

   View > Channel Filter



or select the pop-up menu

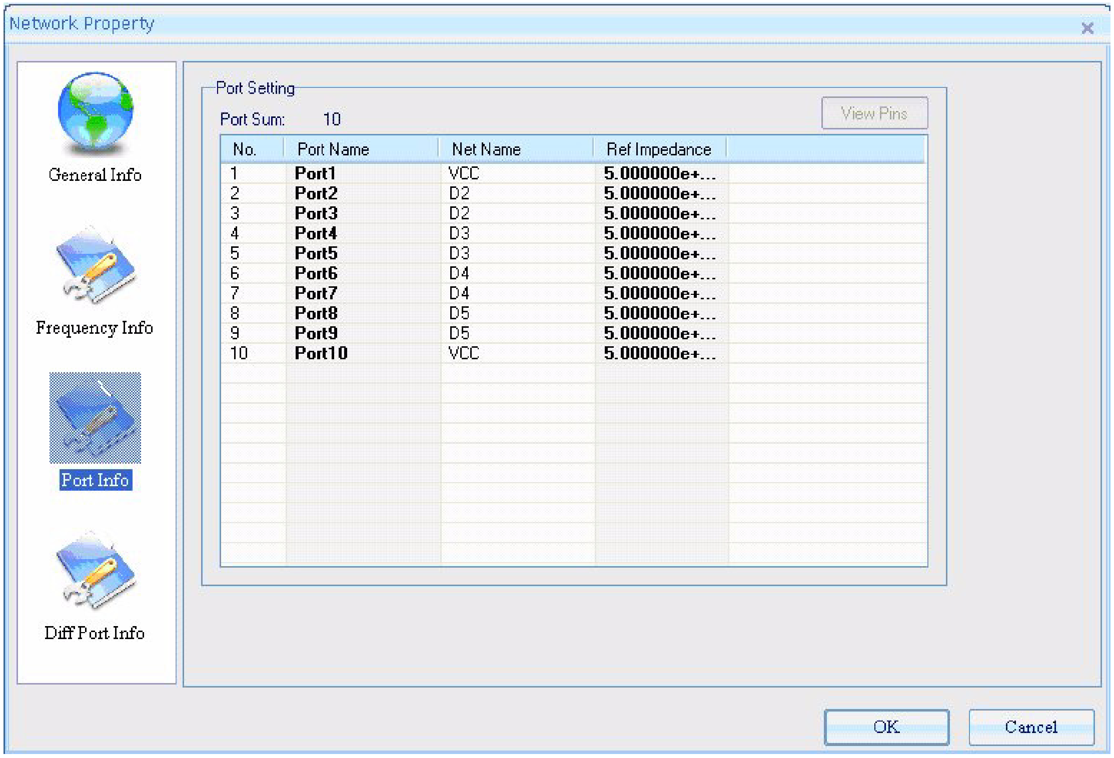
Channel Filter



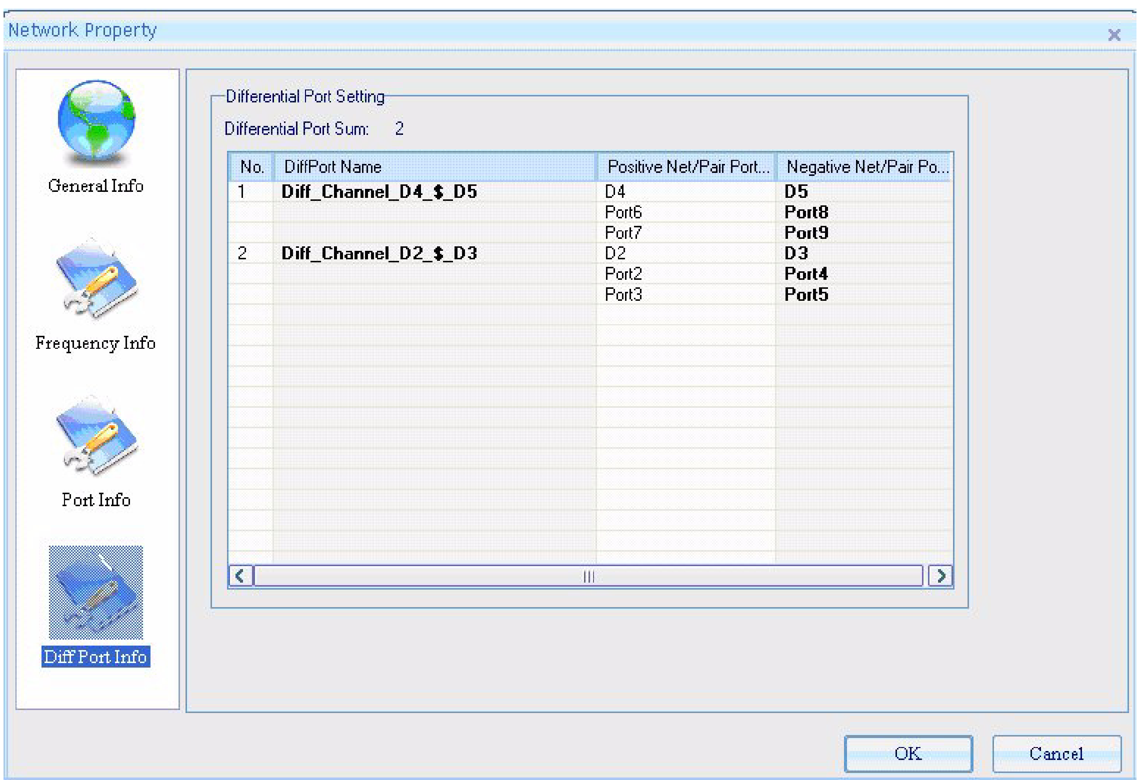
If no network matrix is selected, the **Channel Filter** menu or toolbar button appears gray.

3. A **Channel Filter** dialog appears.
   - For every network, there are 3 views of the matrices items.
   - It depends on the port setting and different channel setting.
   - If there is no different channel setting defined in a network, there is only one view of a single channel matrix.

Port Setting Example



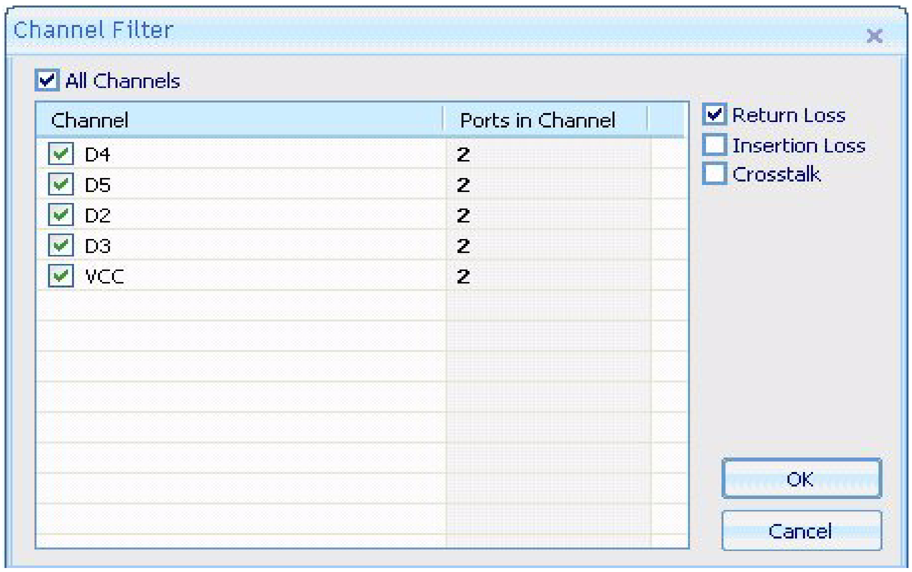
Differential Port Setting Dialog



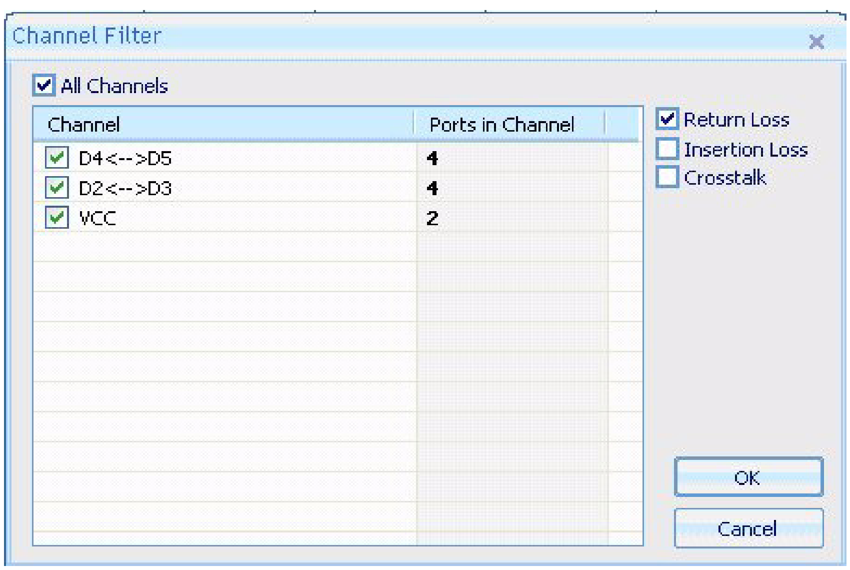
There are three Channel Filter dialogs.

- Mixed Mode Matrix in Differential Channel View.
- Mixed Mode Matrix in Normal View.
- Single Channel Matrix in Normal View.

Single Channel Matrix in Normal View



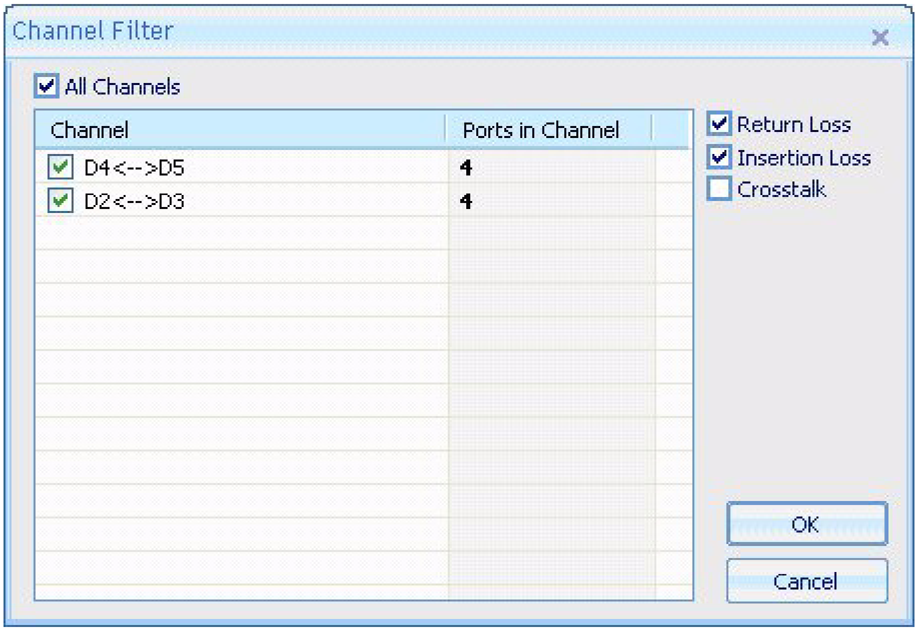
Mixed Mode Matrix in Normal View



Mixed Mode Matrix in Differential Channel View

The dialog below lists all the channels in current matrix. Users can select the channels in the list by a multiple-select method. To select all channels select

All Channels



- **Mixed Mode Network Parameters** — A channel is defined as all the port pairs in a differential net pair.
- **Single-ended Network Parameters** — A channel is defined as all the ports in a net.

Once a channel is selected, all ports under this channel are selected. The sub-matrix corresponding to all the selected ports are used for further selection. The elements in the selected sub-matrix are divided into three categories:

- **Crosstalk** – Rest of the elements.
- **Insertion Loss** – Off diagonal elements whose two ports are in the same channel.
- **Return Loss** – Diagonal elements.

The **Insertion Loss** and **Crosstalk** elements consider the elements in the lower triangular matrix. This dialog can remember last user's selection including:

- Channels
- Crosstalk
- Insertion Loss
- Return Loss

1. Select channels.
2. Check and uncheck the three options on the right side of the pane.
3. Click **OK**.

The displayed elements in the selected network should be replaced by the elements specified in the user selection.

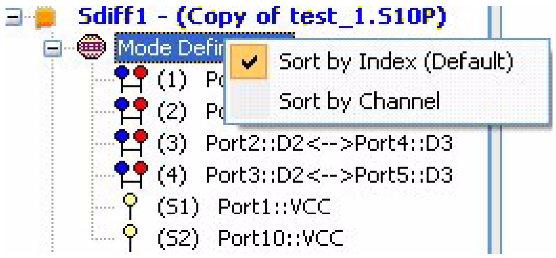
New Organization of Mixed Mode Network Parameters

Mode Definitions under the mixed mode network matrix lists all port pairs of differential channel setting and single channel ports. Single channel ports are only listed in normal view.

This tree item is collapsed by default.

There are two pop-up menus. Use these menus to determine the sorting of the port pairs and single channel ports.

- Mode Definitions > Sort by Index (Default)
- Sort by Channel

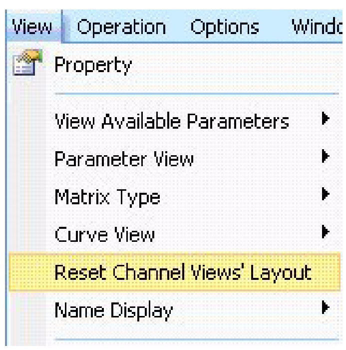


Four curve views are available in the differential channel view pane.

1. Double-click in a curve view to maximize this curve view.
2. Double-click again to restore the size.
3. To restore the default layout of the four curve views click

   View -> Reset Channel Views’ Layout

4. Double-click in the differential channel view to maximize and restore a curve window.



View Mixed-mode Network Parameters

The mixed mode parameters can be viewed in the normal view with the single-ended network parameters or in the differential channel view, which shows only the differential channels. The two views can be toggled by selecting

View > Parameter View > Normal View

or

View > Parameter View > Differential Channel View

This can also be done with the combo box in the **Curve Settings** toolbar. If a loaded network contains differential pair definitions, but the mixed-mode network parameters are not displayed.

1. Select the matrix.
2. Select

   View > View Available Parameters

   or right-click on the selection.
3. From the context menu select

   View Available Parameters
4. Choose the desired view.

Frequency Truncation

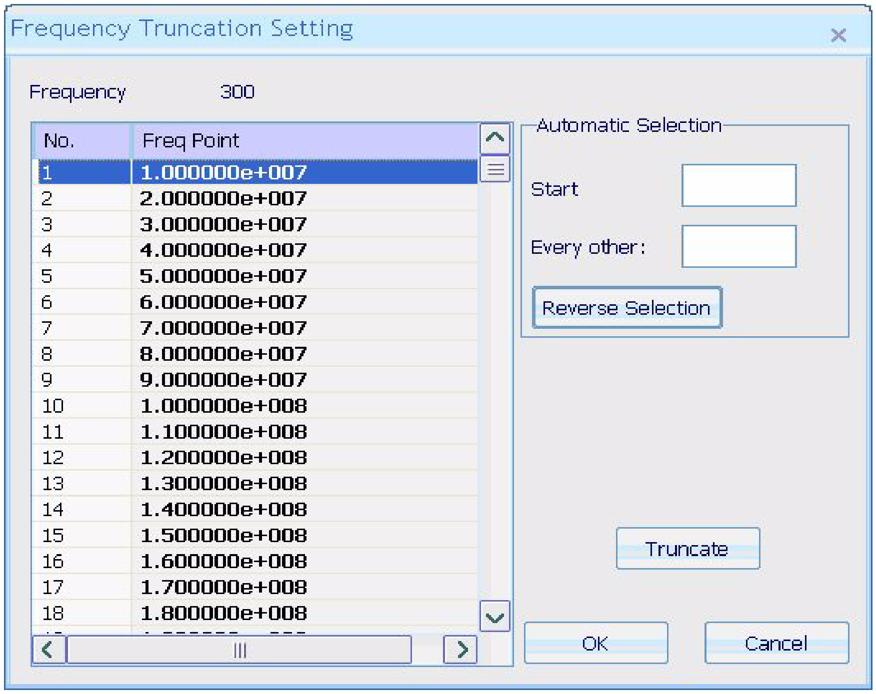
1. Select a network from the Matrix pane.
2. Select

   Operation > Frequency Truncation

   or right-click on the selection.
3. From the context menu, select

   Frequency Truncation
4. Select the frequencies that you want to remove.
5. Click **Truncate** to remove these frequencies.

6. Click **OK** the accept the final frequency sampling. The display is updated.



Frequency Re-sampling

If the network parameters are saved in a BNP file with AFS compression, the sampling frequencies can be changed in the display.

1. Select a network in the matrix pane.
2. Select

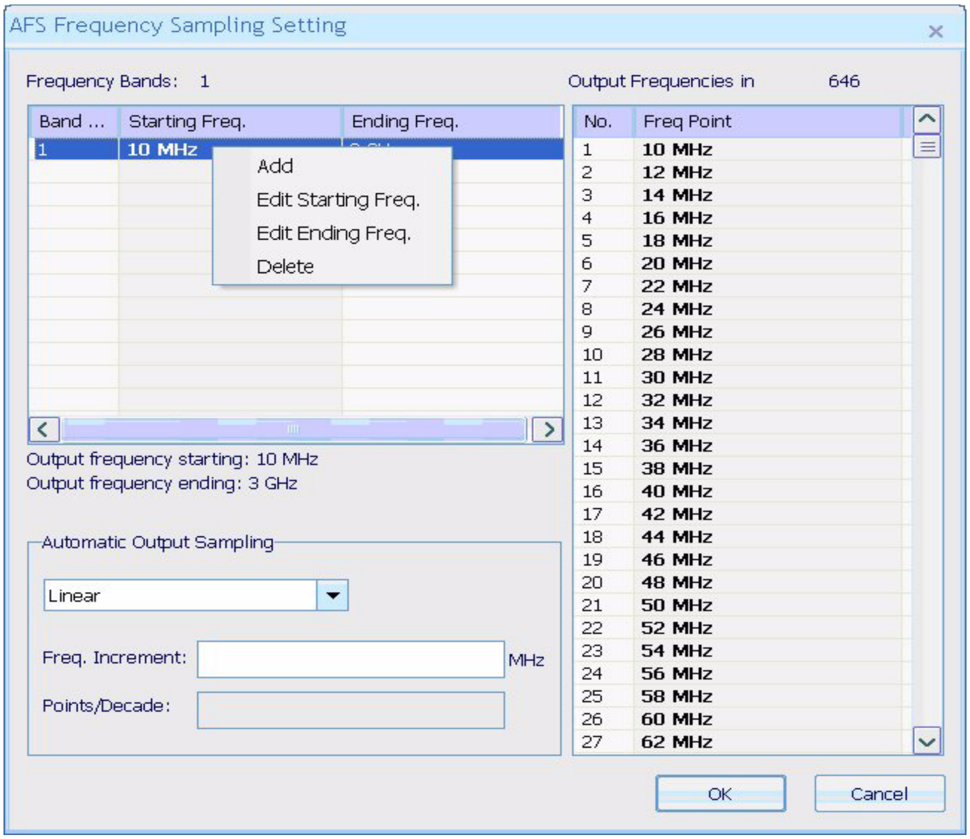
   Operation > Operation > Frequency Resampling

   or right-click on the selection.
3. In the Context menu, select

   Frequency Resampling
4. Right-click to select the frequency band. You can choose to:
   - Add a new band.
   - Delete the band.
   - Edit the ending frequency of the band.
   - Edit the starting frequency of the band.
5. Select a **Sweeping Mode** from the **Automatic Output Sampling** combo box.
6. Modify the sweeping information and click **Return**. The sampling with this band is updated.
7. Select a frequency from the list on the right. Right-click.
8. Edit the value or insert a frequency point above or below the selected frequency.
9. You can select multiples of some frequencies. Right-click.
10. Remove them from the list.

11. Click **OK** the accept the change. The display curves id updated.

NOTE!	The re-sampling is for display purpose. If you save the file in BNP format with AFS compression, the new sampling is not kept. To keep the new sampling, you can save the file in Touchstone format or discretely sampled BNP format.



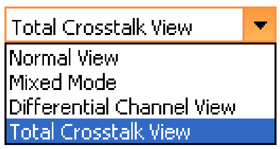
View Total Crosstalk of Network Parameter

NOTE!	**Total Crosstalk View** is only available on reloaded result, not for current simulated result.

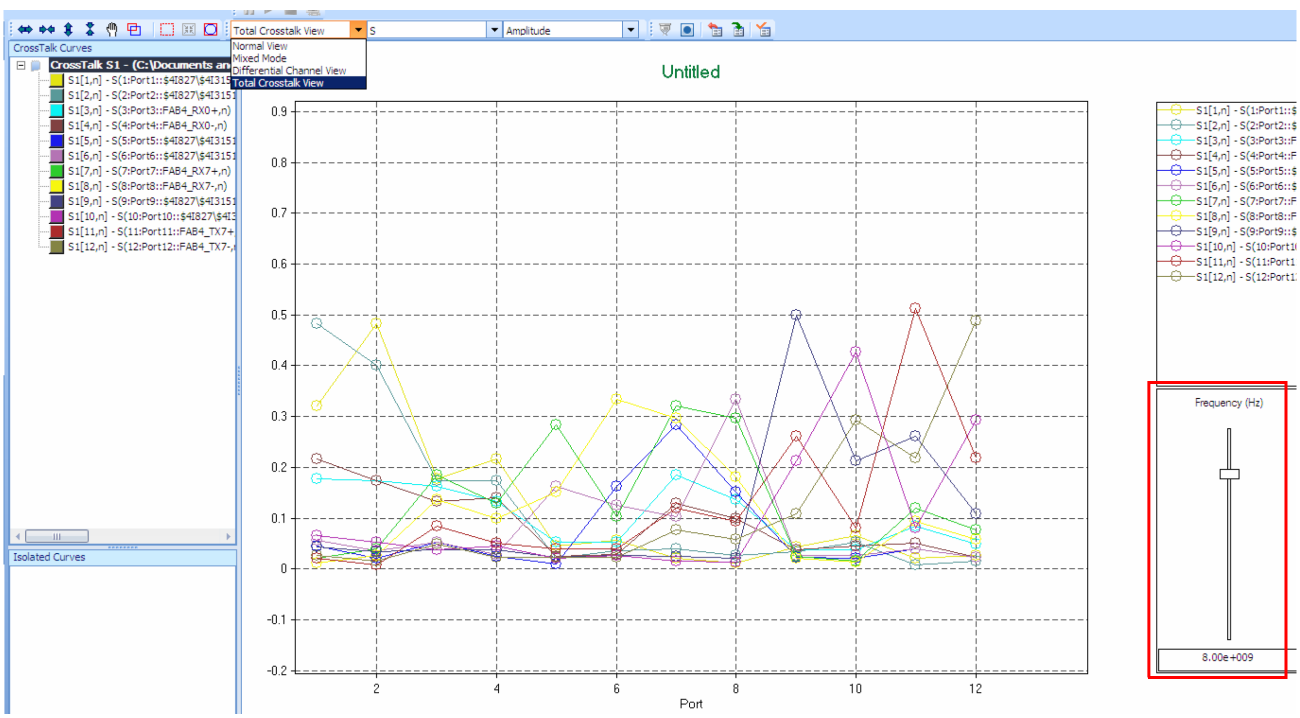
To view **Total Crosstalk View**, select

View > Parameter View > Total Crosstalk View.

Or select **Total Crosstalk View** from the toolbar.



The crosstalk result is displayed in the curve pane.



- Each curve shows the value of crosstalk from S[i,1] to S[i,n].
- The X axis shows the port number from 1 to n.
- The Y axis represents the S/Y/Z Parameter.
- To view the crosstalk at desired frequency, you can use the scroll bar or enter specific number at the right-bottom corner.

Curve Operations

Load / Save Isolated Curves in PowerSI

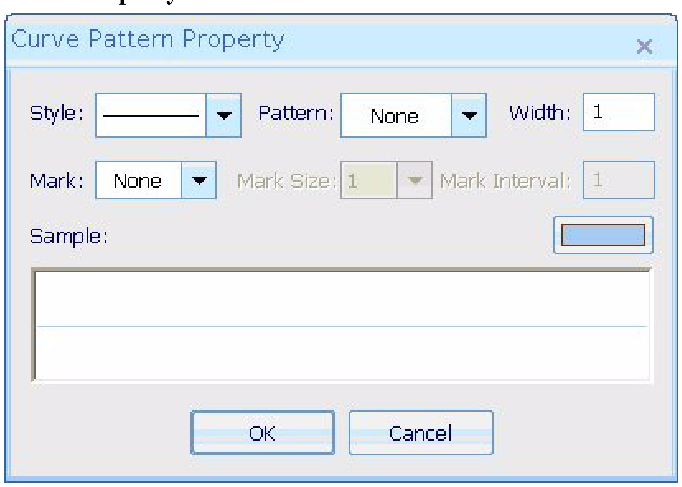
1. Right-click in the **Isolated Curve** pane.
2. Select **Load** from the Context menu.
3. Select the curves in the **Isolated Curve** pane to save the curves.
4. Right-click on the selection.
5. Select **Save** from the context menu.

View and Set Curve Properties

1. Select one or more matrix elements or one or more isolated curves.
2. Select

   View > Property

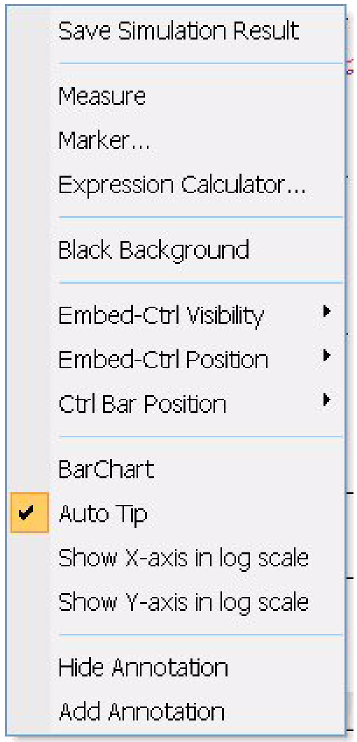
   or right-click on the selected curves.
3. Select **Property** from the context menu.



4. You can edit the following properties in the **Curve Property** dialog:
   - Curve Color
   - Line Pattern
   - Line Style
   - Line Width
   - Marker Size
   - Marker Type
5. Click **OK** to accept the changes. The selected curves are updated in the Curve window.

Curve Pane Context Menu

Right-click in the Curve window. A context menu appears.



You can perform the following operations in this menu.

- **Add Annotation** — Add a text string in the Curve window.
- **Auto Tip** — Show / Hide the tip of the objects in the Curve window when moving the mouse.
- **Bar Chart** — Toggle the plot style between a bar chart and a continuous line.
- **Black / White Background** — Set the background of the curve window to be black or white.
- **Ctrl Bar Position** – If a sub window is docked, change the position of the docking.
- **Embed-Ctrl Position** – Toggle the sub windows between floating or docking.
- **Embed-Ctrl Visibility** – Set the visibility of the sub windows (for example, the legend bar) in the display area.
- **Marker** — Toggle the horizontal and vertical marker lines.
- **Measure** — Toggle the horizontal and vertical measure lines.
- **Show X-axis in Log Scale** — Toggle the X axis between log scale and linear scale.
- **Show Y-axis in Log Scale** — Toggle the Y axis between log scale and linear scale, available only if the value type is set to Amplitude.

Copy Curve

Curve Copy is a simple way to compare the network parameters of different types. The isolated curves are always the same regardless of the parameter matrix type.

1. Select one or more matrix elements, or one or more isolated curves.
2. Select

   Operation > Copy

   or right-click.
3. Select **Copy** from the context menu. The duplicates of the selected curves are added to the **Isolated Curve** pane.

Curve Calculation

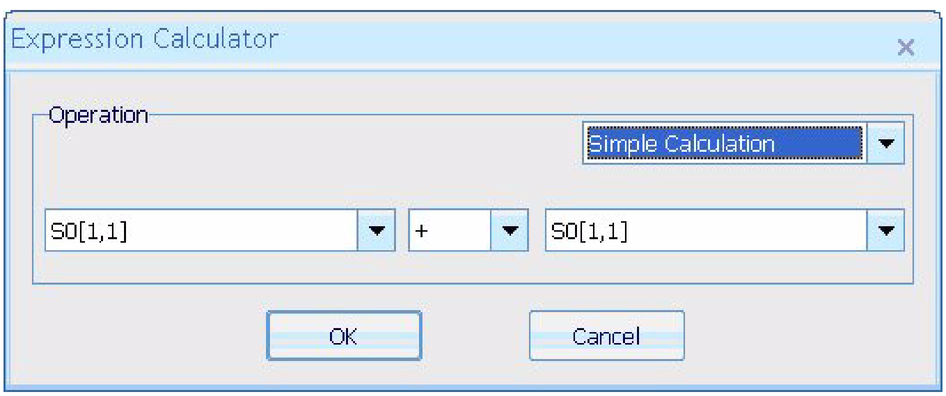
1. Select

   Operation > Expression Calculator

   or right-click in the curve window.
2. In the context menu, select

   Expression Calculator

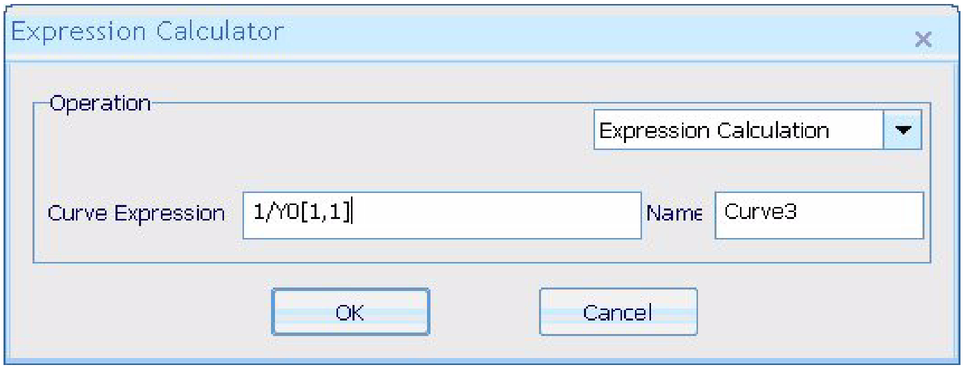
   You can select the displayed curves or constants from the combo box and create a simple expression.

   In the **Expression Calculation mode**, you can enter an expression with the matrix element unique ID or the isolated curve names.

   A unique curve name is required for the new curve, and a default name is provided. The matrix elements can be anything that is available; it is not limited to the displayed elements.

3. Click **OK**. The calculation result is added as a new isolated curve.



Expression Customization

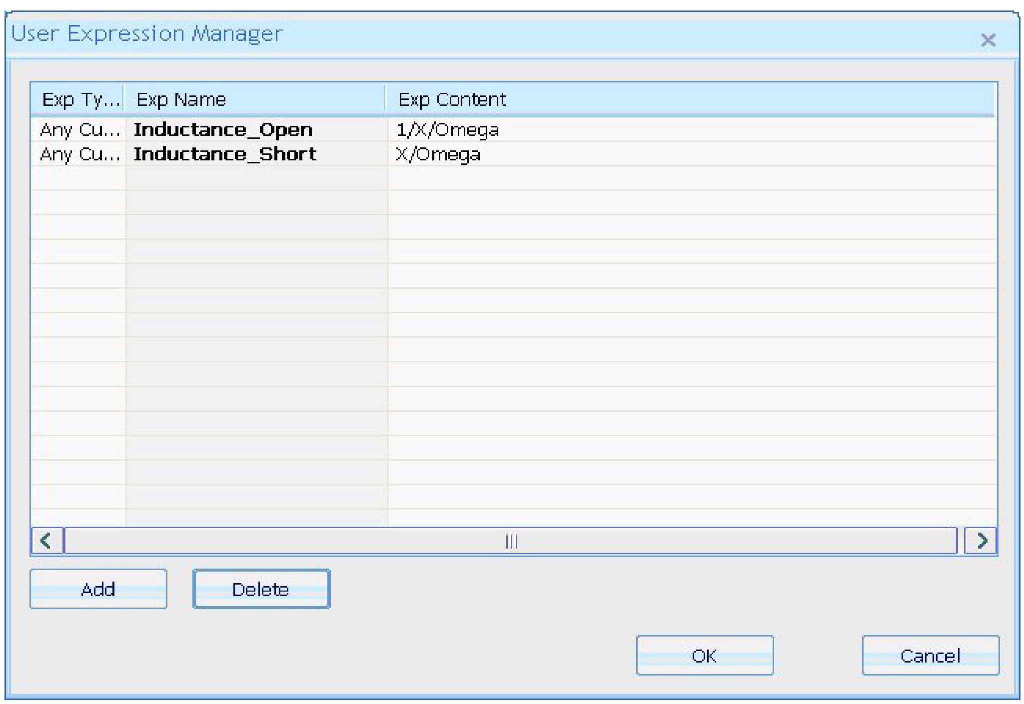
You can re-use an expression in future calculations.

1. Create a curve using Expression Calculation.
2. Select the calculated curve.
3. Right-click on the selection.
4. In the context menu, select

   Use Expression

   The Expression Manager is displayed.

   You can edit the name and contents of the expressions for later user. The symbol **X** represents the curve in the calculation.
5. Click **OK** to save the list of expressions.



Use a Saved Expression

1. Select one or more matrix elements or one or more isolated curves.
2. Right-click on the selection.
3. From the context menu select

   Calc Curves by Expression

4. In the Expression Manager select an expression.
5. Click **OK**. The expression is applied to all the selected curves. The calculation result is shown as new curves.

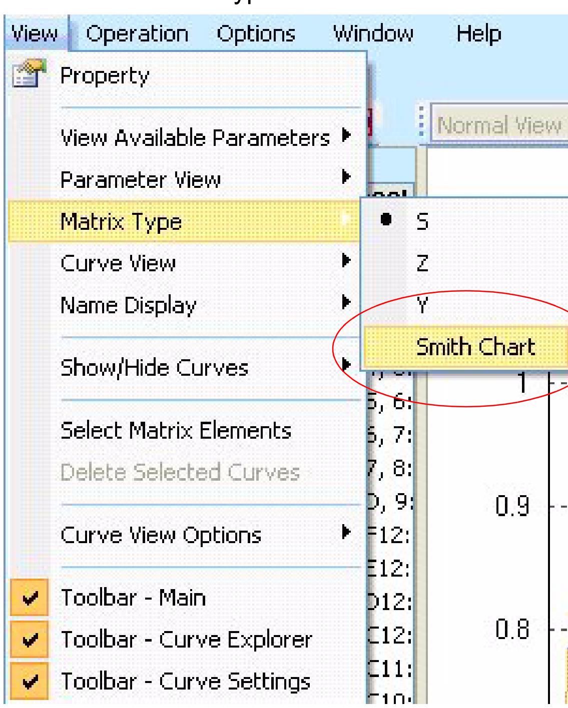
NOTE!	The validity of the expression depends on the parameter matrix and the port terminations.

SMITH CHART VIEW OF THE S PARAMETER

Select **Smith Chart** from the Curve Source combo box;

or select Smith Chart from the menu

View > Matrix Type > Smith Chart

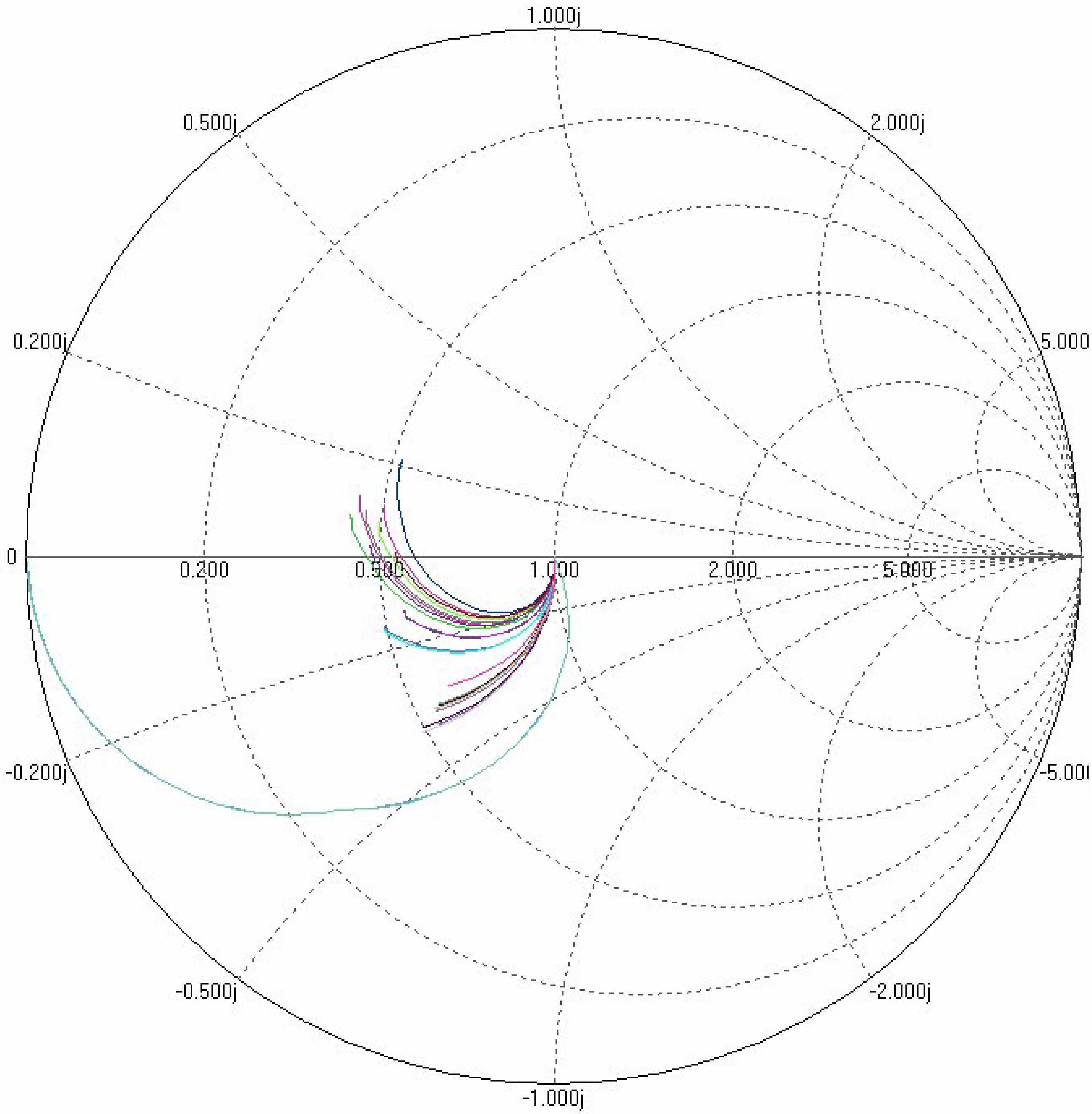


Overview

Use the toolbar to view the Smith Chart.

- Enlarge Reduce X / Y-scale buttons are disabled.
- Curve Method Selection combo box is disabled.

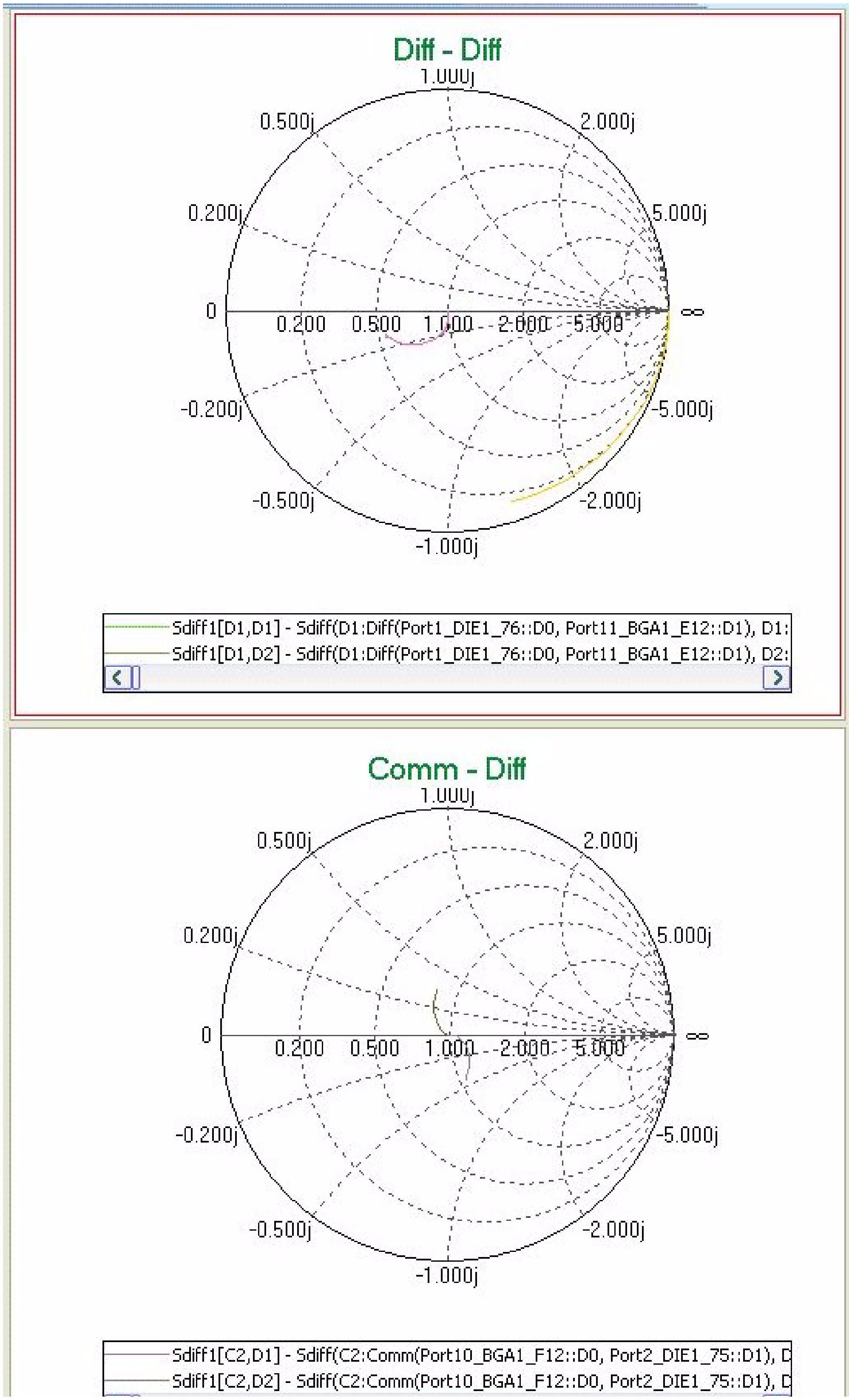
The Smith Chart in **Normal view** is shown in the following illustration.

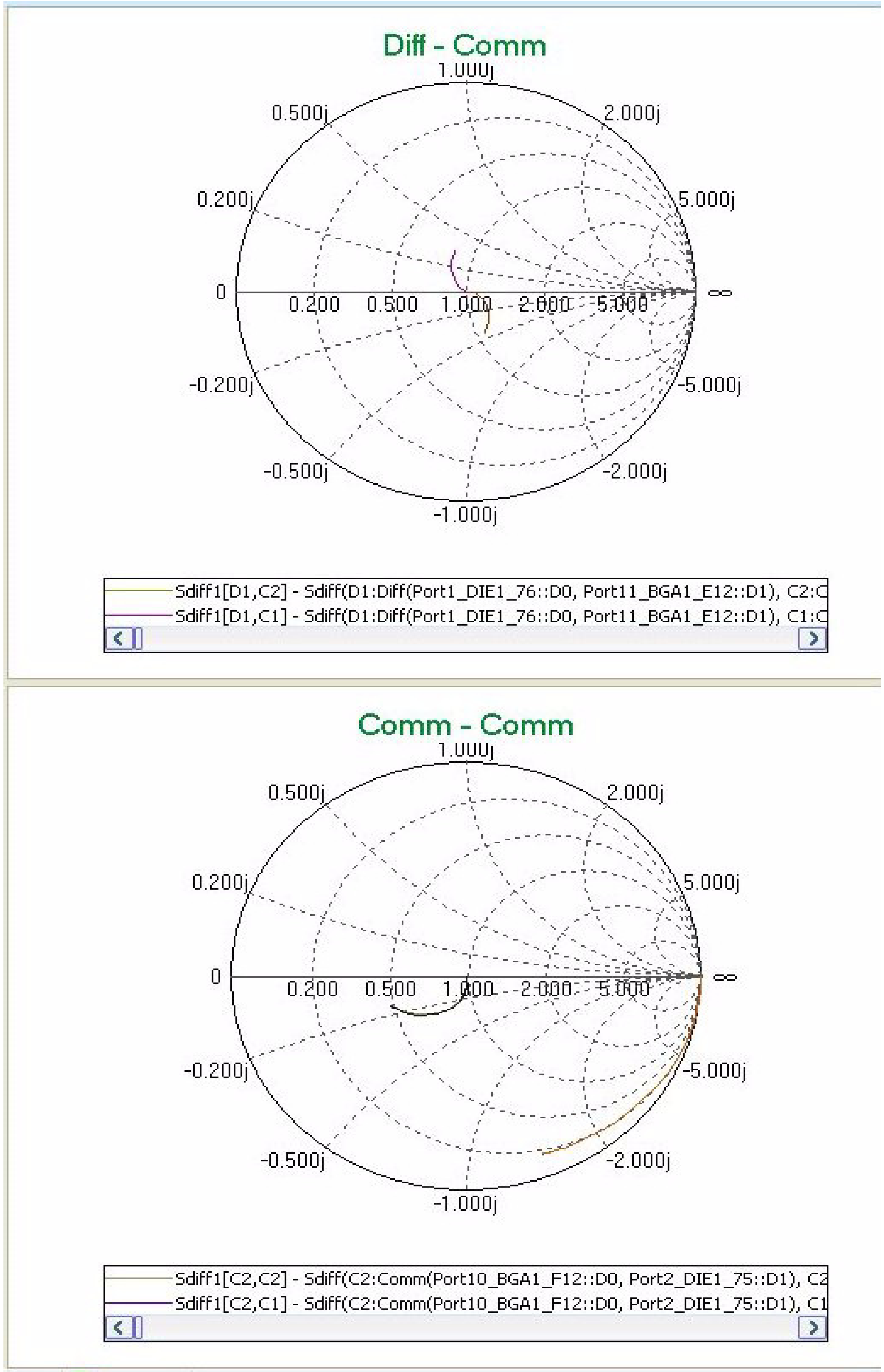


Differential Channel View

The Smith Chart plot in **Differential Channel View** is shown in the following illustrations.

- The first set shows **Diff-Diff** and **Comm-Diff**.
- The second set shows **Diff-Comm** and **Comm-Comm**.


Diff - Diff
1.000j
0.500j
2.000j
0.200j
5.000j
0
0.200
0.500
1.000
2.000
5.000
∞
-0.200j
-5.000j
-0.500j
-2.000j
-1.000j
Sdiff1[D1,D1] - Sdiff(D1:Diff(Port1_DIE1_76::D0, Port11_BGA1_E12::D1), D1:
Sdiff1[D1,D2] - Sdiff(D1:Diff(Port1_DIE1_76::D0, Port11_BGA1_E12::D1), D2:
Comm - Diff
1.000j
0.500j
2.000j
0.200j
5.000j
0
0.200
0.500
1.000
2.000
5.000
∞
-0.200j
-5.000j
-0.500j
-2.000j
-1.000j
Sdiff1[C2,D1] - Sdiff(C2:Comm(Port10_BGA1_F12::D0, Port2_DIE1_75::D1), D
Sdiff1[C2,D2] - Sdiff(C2:Comm(Port10_BGA1_F12::D0, Port2_DIE1_75::D1), D

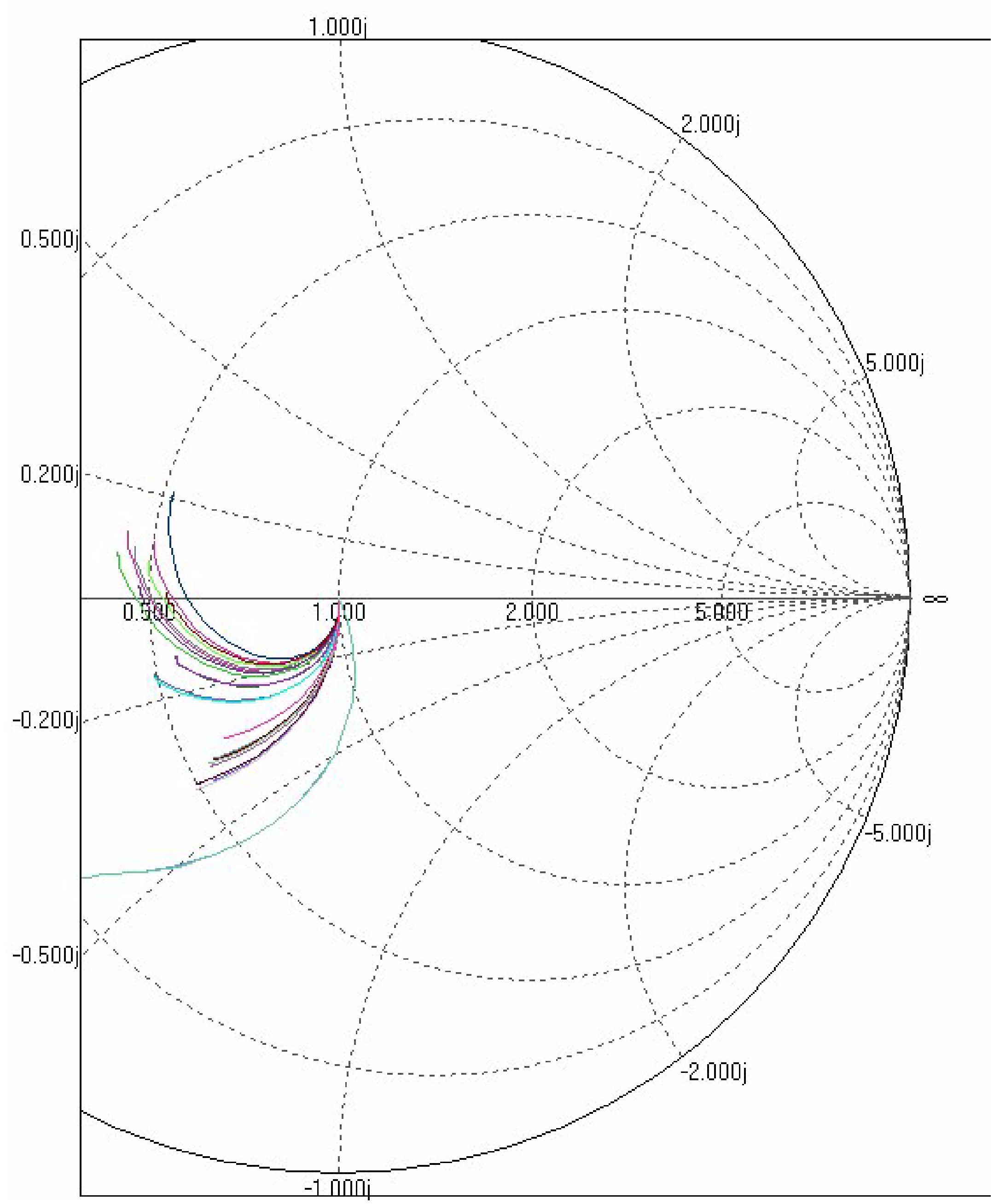

Diff - Comm
1.000j
0.500j
2.000j
0.200j
5.000j
0
0.200
0.500
1.000
2.000
5.000
∞
-0.200j
-5.000j
-0.500j
-2.000j
-1.000j
Sdiff1[D1,C2] - Sdiff(D1:Diff(Port1_DIE1_76::D0, Port11_BGA1_E12::D1), C2:C
Sdiff1[D1,C1] - Sdiff(D1:Diff(Port1_DIE1_76::D0, Port11_BGA1_E12::D1), C1:C
Comm - Comm
1.000j
0.500j
2.000j
0.200j
5.000j
0
0.200
0.500
1.000
2.000
5.000
∞
-0.200j
-5.000j
-0.500j
-2.000j
-1.000j
Sdiff1[C2,C2] - Sdiff(C2:Comm(Port10_BGA1_F12::D0, Port2_DIE1_75::D1), C2
Sdiff1[C2,C1] - Sdiff(C2:Comm(Port10_BGA1_F12::D0, Port2_DIE1_75::D1), C1

Manage the Curve View

The default plot display an unit circle, grid lines and curves. If the unit circle can be seen entirely, the square disappears; otherwise, it appears.

Move by Pan

1. Click the button  .
2. Press the plot.
3. Move to the desired position. Release the mouse button.



Zoom In and Out

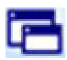
Use the mouse wheel in a valid area to **Zoom** in and out of the plot to see the interested area.The grid line density of the Smith Chart automatically adjusts according to the Zoom status.

Area Select / Zoom Back

1. Click the button 

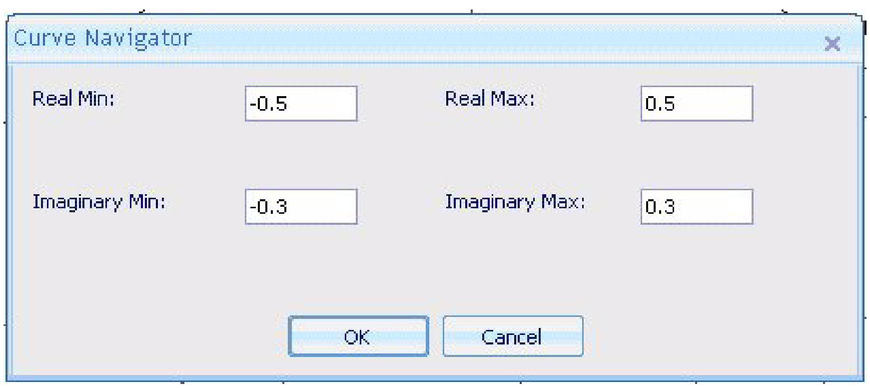
.
2. Select a rectangle in valid area to stand out the interested area.
3. The grid line density of the Smith Chart automatically adjusts according to the Zoom status.
4. Click the  button to **Zoom** back to the last area selected status.

Fit

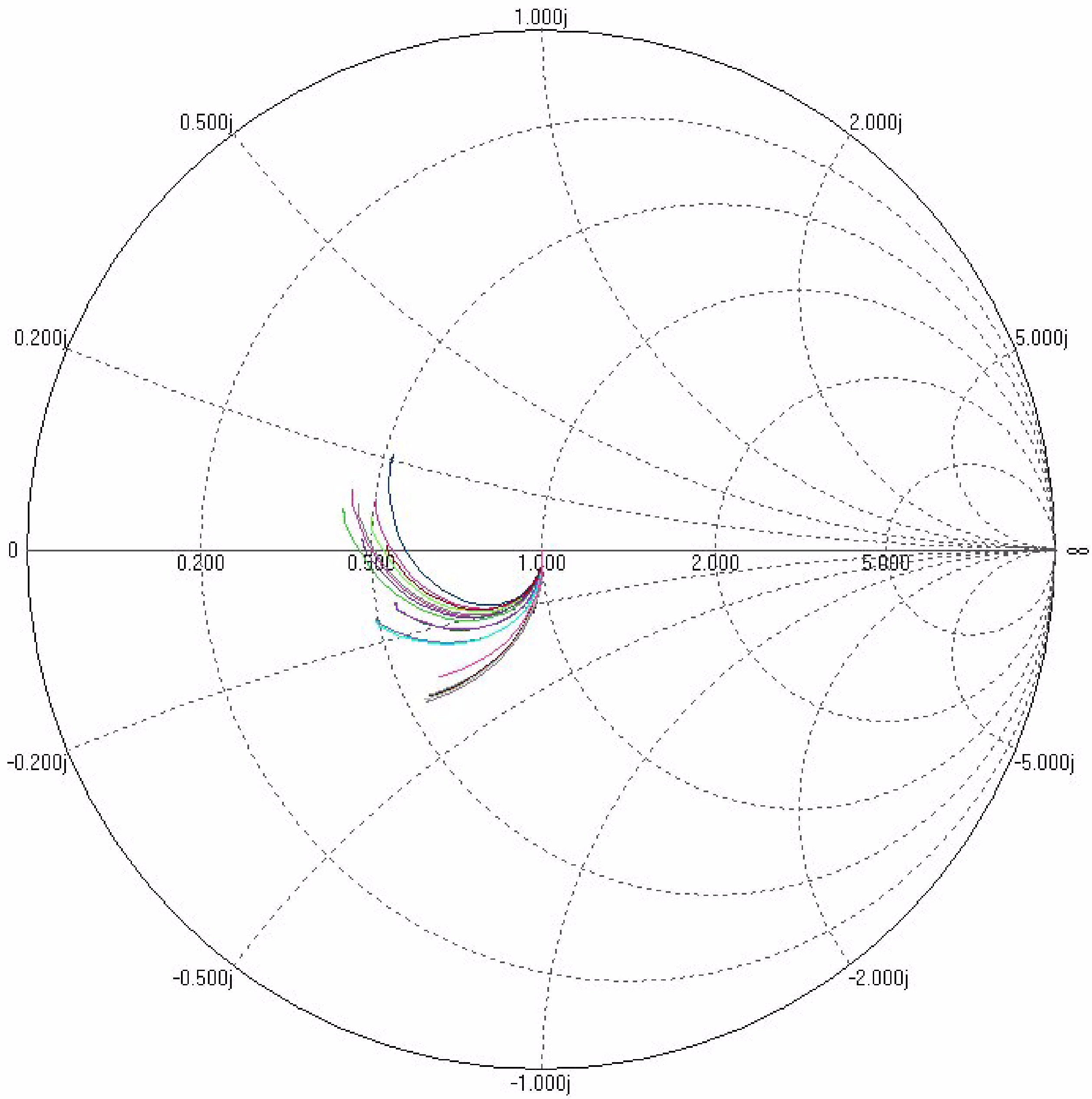
Click the  button to bring the plot to the default setting.

Navigator

1. Open a dialog.
2. Input the **Real Min** and **Real Max** values.
3. Click **OK**.

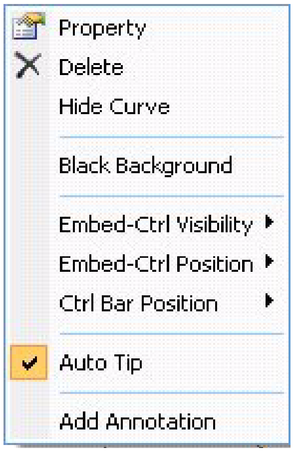


The plot is redrawn according to the new settings.



Menu Functionality

1. Move the mouse on a curve.
2. Right-click. The menu appears.



- **Add Annotation** — Click **Add Annotation** to switch the edit status to adding annotation.
- **Auto Tip** — Click **Auto Tip** to toggle the option of showing tip.
- **Black Background** — Click **Black Background** to toggle the background of the plot.
- **Delete** — Click **Delete** to delete the selected curve.
- **Embed-Ctrl Visibility / Position** — Float or resize the legend bar or rearrange the fixed position.
- **Hide Curve** — Click **Hide Curve** to hide the selected curve.
- **Property** — Click **Property** to set up the curve pattern of the selected curve.

Auto-tip

There are three sorts of tip in Smith Chart view:

- X-Axis
- Y-Axis
- Curve node

The autotip of the Curve node includes:

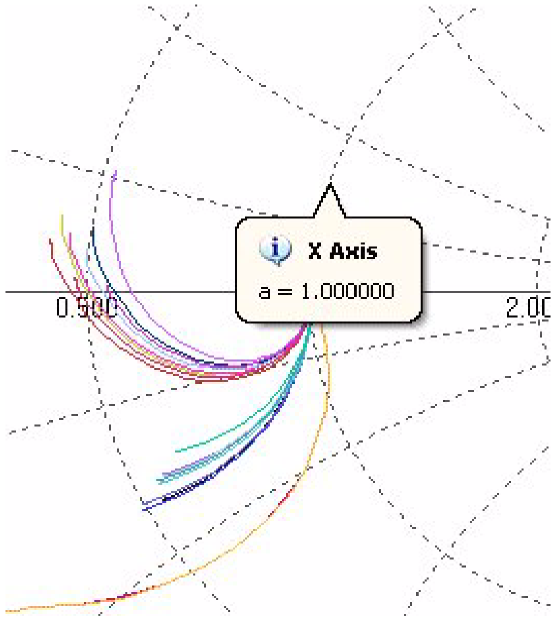
- Complex Value of the S Matrix Element at this Frequency.
- Frequency.
- Name of the S Matrix Element.
- Normalized Admittance Y.
- Normalized Impedance Z.
- VSWR.

Given the **S matrix** element value as s, the read-out is:

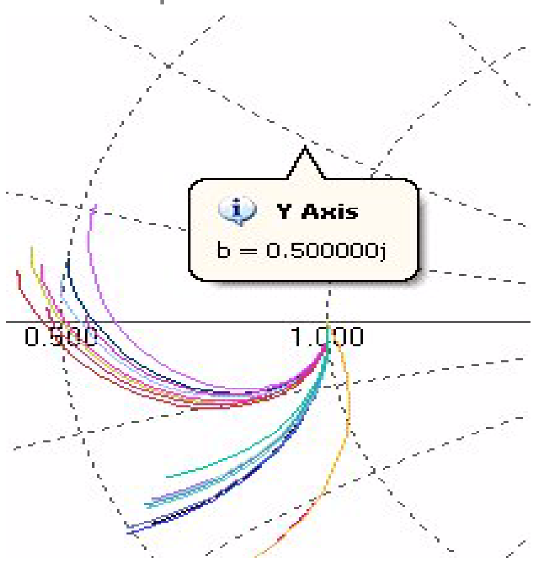
$$z = \frac{1+s}{1-s} \qquad VSWR = \frac{1+|s|}{1-|s|}$$

$$y = \frac{1-s}{1+s}$$

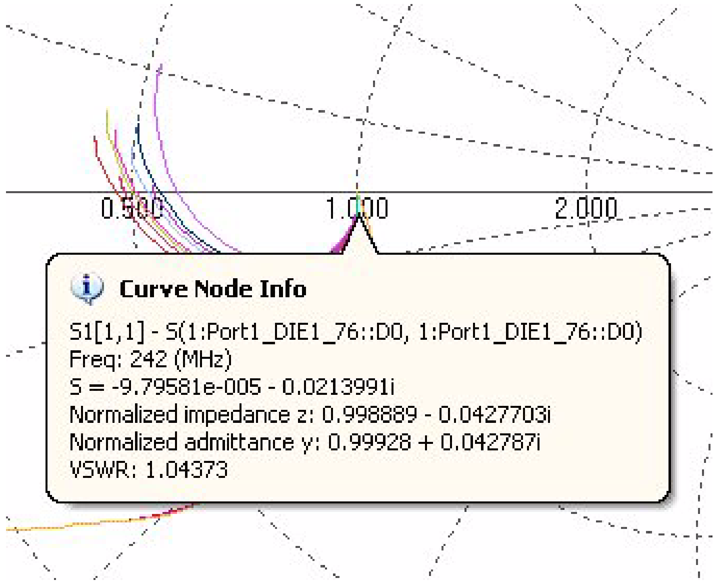
X-Axis Example



Y-Axis Example



Curve Node Info



Chapter

11

Customize Workflow

This chapter introduces the customized workflow and takes you through customization process step-by-step.

Lesson One: Understanding the Default Workflow

The default workflow now includes a customization option.

1. Click:

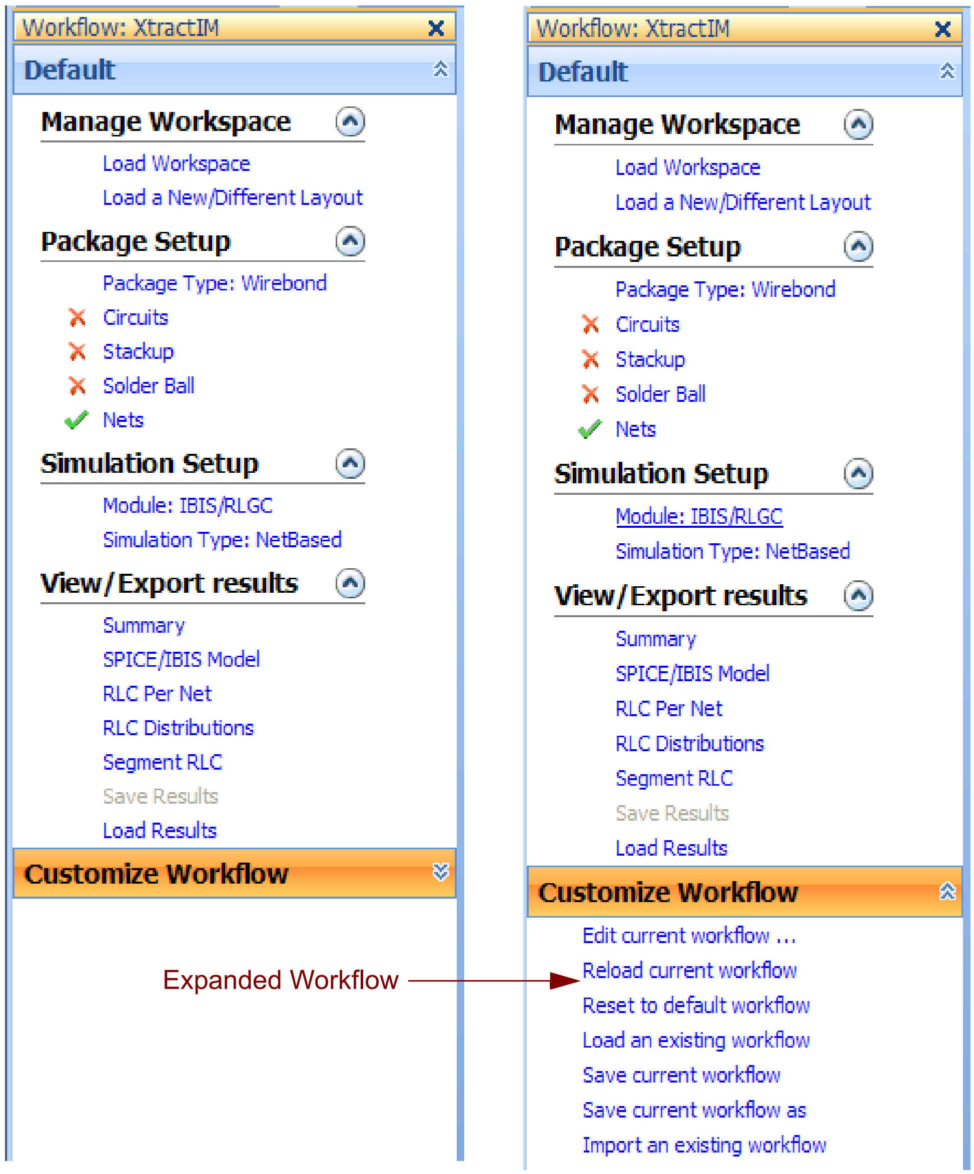
   Customize Workflow

   The Customize Workflow section in the Workflow expands and displays the customization options.

2. Click:

   Edit current workflow

The Current workflow can be customized.



LESSON TWO: EDIT WORKFLOW

You can add, rename and change the sequence of the Workflow.

1. Click:

   Simulation Setup > New

2. Click:

   New Scheme

3. Change the name from **New Step** to **Extraction Frequency**.
4. Click **Set**.
5. Click:

   Setup > Extraction Frequency

6. Click **OK**.
7. Click:

   Package Type > Circuits

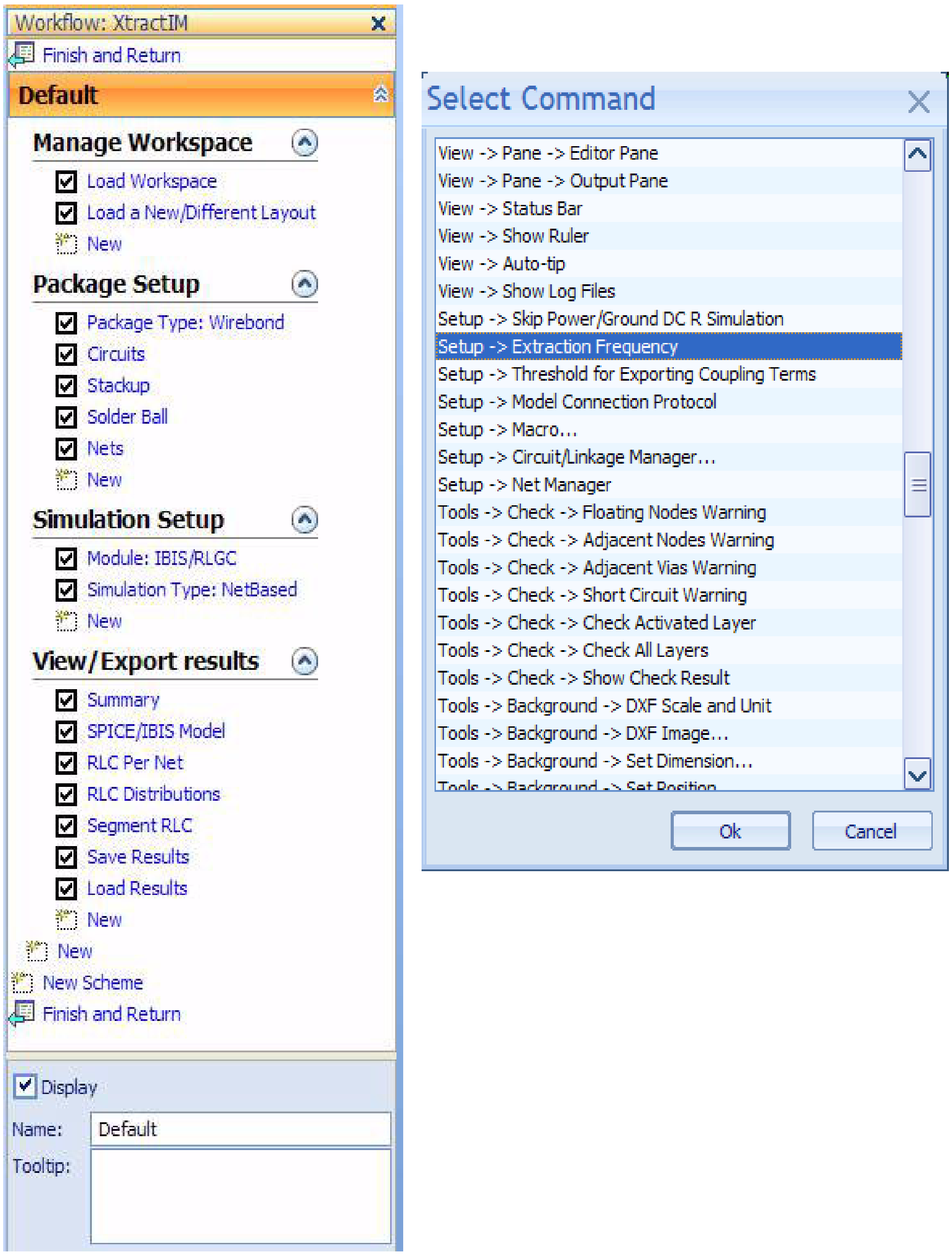
8. Change the name **Circuits** to a name your prefer (such as **Circuit Setup**).
9. In the Workflow pane, move **Nets** to:

   Package Type > Wirebond

10. Click:

Finish and Return

The new **Workflow** is updated.



Lesson Three: Save the Workflow

1. Click:

   Customize Workflow > Save Current Workflow

   An XtractIM.cfg file is saved in the same directory as XtractIM.exe.

2. Click:

   Customize Workflow > Reset to default workflow

   The workflow goes to **Default Workflow**.

Load Existing Workflow

1. After launching XtractIM, the saved XtractIM.cfg is automatically launched.

   The customized workflow is loaded.

2. Select:

   Customize Workflow > Reset to default workflow

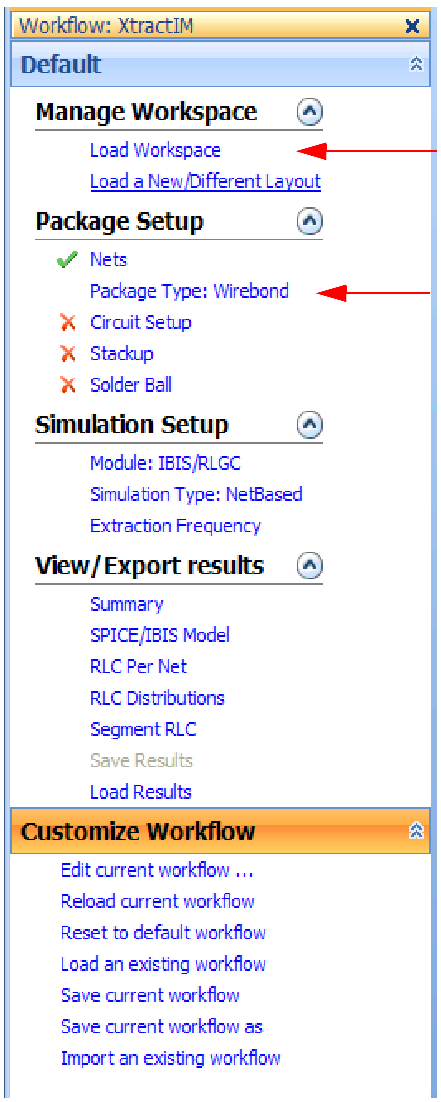
   The workflow goes to Default Workflow.

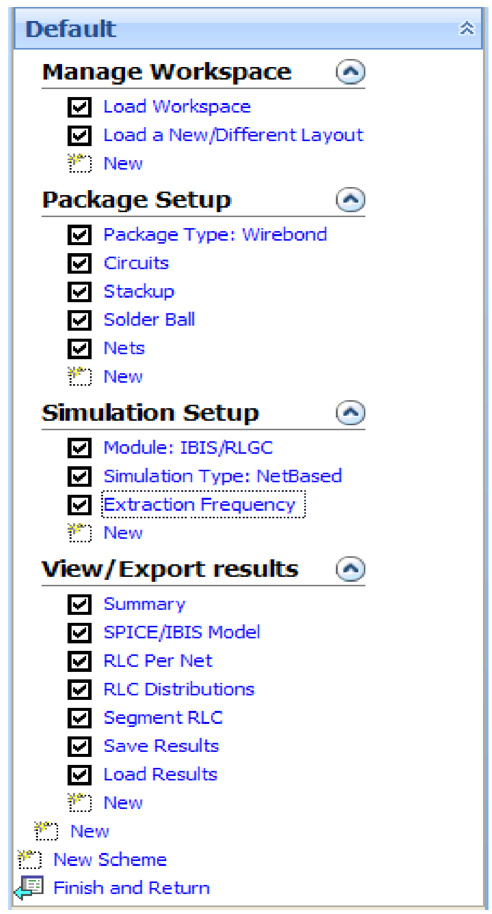
3. To substitute the current file, select:

   Simulation Setup > Extraction Frequency

   or select:

   Import an existing workflow *.cfg file





Index

Symbols

Numerics

A

B

C

D

E

M

N

O

S