Simyog Technology Pvt. Ltd.

presents

Compliance Scope

Virtual Laboratory for EMI/EMC

User Guide

Created: 8th Feb 2018

Last Modified: 21st May 2019 (Release 2.2.7)

Table of Contents

1 Chapter 1: Introduction to Compliance Scope

1.1 What is Compliance Scope (CompScope)

CompScope, is a Virtual Electromagnetic Interference/ Electromagnetic Compatibility (EMI/EMC) laboratory where designers can validate and improve their hardware at an early stage by uploading their design files. The tool also provides diagnosis and suggestions for cost-effective fixes at the Printed Circuit Board level.

Electromagnetic emissions within an automotive, especially electric vehicles, is a key-challenge for automotive IC suppliers, automotive electronic component makers and OEMs. Currently, the problem is addressed by stringent EMC compliance requirements. From the product development side, this is addressed at a late-stage post product prototype, in an EMI/EMC laboratory. Most of the time, the fix requires additional bill of materials in the form of filters or decoupling capacitors.

Compliance-Scope not only allows the designer to predict the EMI/EMC performance by uploading the early design data, but also provides diagnosis and suggestions for meeting specifications. This saves valuable time-to-market owing to reduced sample spins and reduced bill-of-materials by precluding late stage addition of shielding and noise suppressing components.

Any automotive electronic product has to go through a set of four major emission and immunity test certifications for it to be deemed as compliant and hence fit for use in a noisy electromagnetic environment. The test methods are – Bulk Current Injection (BCI), Radiated Emission (RE), Radiated Immunity (RI) and Conducted Emission (CE). Compliance Scope v.2.1.1 supports BCI and RE test methods. The other test methods will be added soon in future releases of CompScope.

1.2 Technology – Under the hood

The tool provides a single solution platform thereby eliminating the usage of multiple solvers and cross-solver integrations. CompScope presents a conglomerate of different types of solvers:

- 3D Full-Wave Electromagnetic solver for PCBs, connectors, housing
- 2D solver for cables
- Open source circuit simulator NGSPICE

The technology used in our product includes:

a) Physics based electromagnetic analysis
b) Learning- based algorithms
c) Matrix compression technology
d) OpenMP based Parallelization strategy

For further details on the technology used in our products please contact info@simyog.com.

1.3 Target Users and Value Proposition

The target users for CompScope are:

a) Automotive electronics Hardware Engineers

b) Automotive electronics EMC Simulation Engineers

c) Automotive electronics IC engineers

d) Possible use in Aerospace, Medical and Consumer Electronics

The goal of the product is to:

- Empower hardware design engineers with simulation capability
- Enable EMC compliance at an early design stage
- Reduce design spins from concept to production
- Figure of merit analysis of any EMI/EMC mitigating components

1.4 Input/Output Parameters for CompScope-BCI and RE

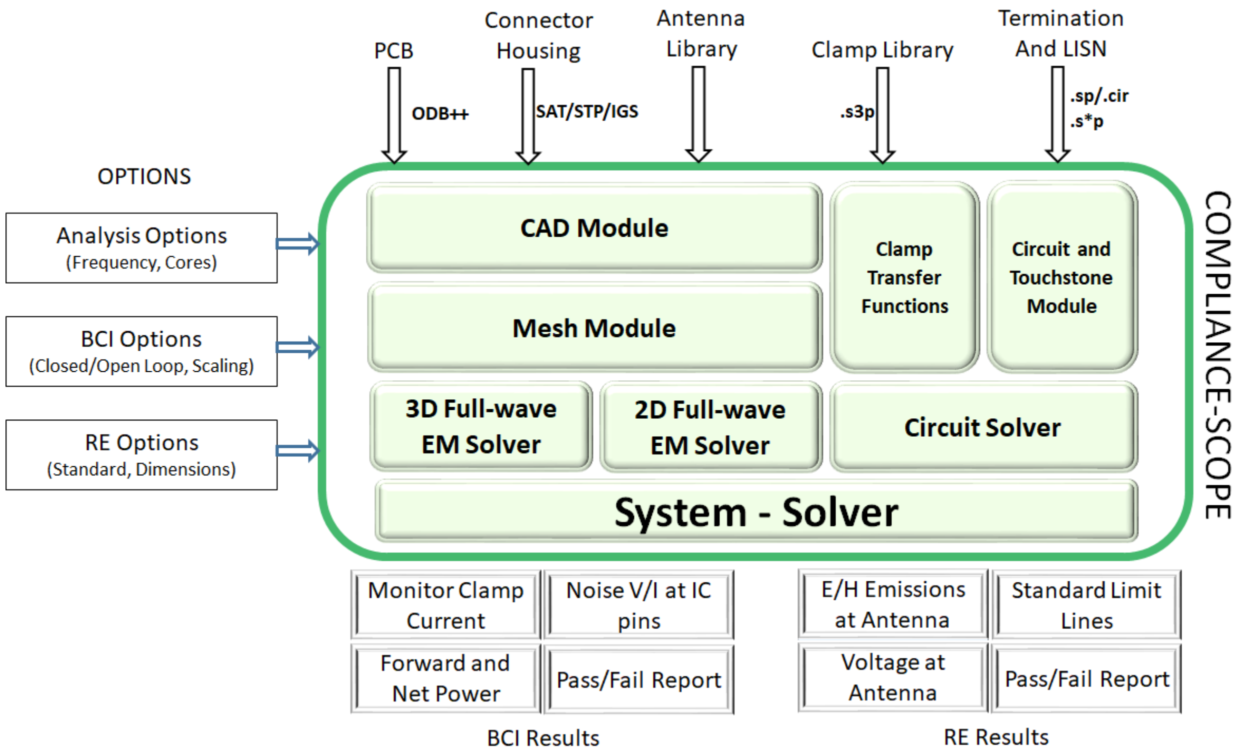


Figure 1-1: Inputs/Outputs for CompScope – BCI and RE

2 Chapter 2: Installation

This chapter describes the process to install the Compliance Scope tool and get it started on a laptop/PC/server.

2.1 Platforms Supported

- Windows 7/8/10 editions x64
- Linux version: Coming soon

2.2 System Requirements

- Recommended more than 64GB Hard Disk Space
- Recommended more than 4GB RAM
- Works best with display resolution of 1280 x 720 or above.

2.3 Licensing

CompScope supports the following license types:

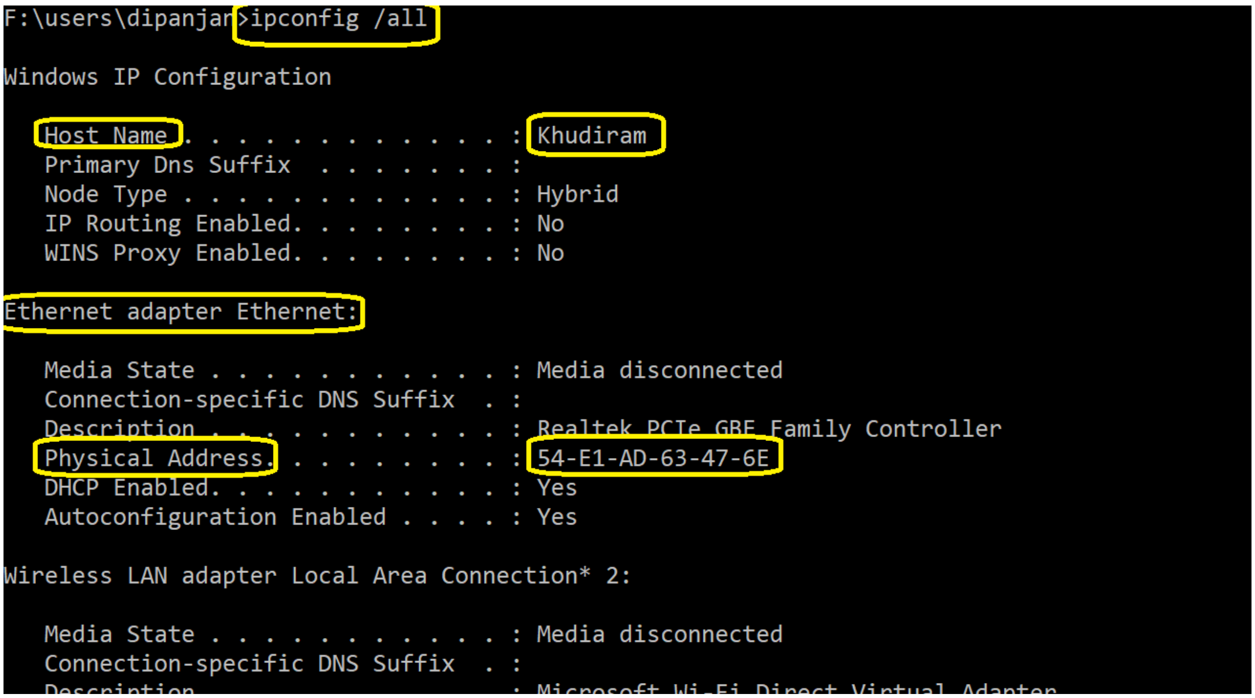
Table 1: Licensing schemes for CompScope

License Type	Feature	License Generation	User action on receiving license
Node-locked	CompScope can only run on the desired machine whose physical address was provided for license generation	Send the physical address of the desired machine to Simyog	1. Start CompScope 2. Select "specify the license file" 3. Browse to the license file
Floating	Any machine with access to the license server machine	Send the hostname and Ethernet physical address to Simyog for license creation	1. Server side: Setup the license server with the license file (explained in 2.3.1) 2. Client side: (a) Start CompScope (b) Select "specify the license server system"

2.3.1 Floating License

2.3.1.1 Server Side

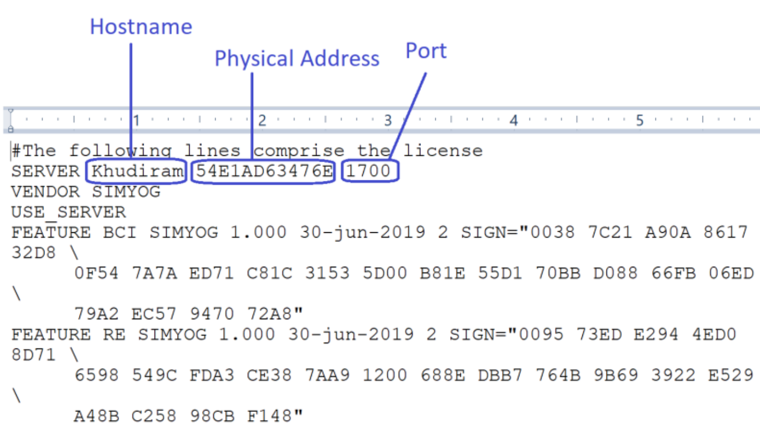
Step 1: Provide the hostname and physical address of the desired license server machine to Simyog Applications Team. This can be done by typing “ipconfig /all” on the windows command line.



```
F:\users\dipanjan>ipconfig /all

Windows IP Configuration

   Host Name . . . . . . . . . . . . : Khudiram
   Primary Dns Suffix  . . . . . . . :
   Node Type . . . . . . . . . . . . : Hybrid
   IP Routing Enabled. . . . . . . . : No
   WINS Proxy Enabled. . . . . . . . : No

Ethernet adapter Ethernet:

   Media State . . . . . . . . . . . : Media disconnected
   Connection-specific DNS Suffix  . :
   Description . . . . . . . . . . . : Realtek PCIe GBE Family Controller
   Physical Address. . . . . . . . . : 54-E1-AD-63-47-6E
   DHCP Enabled. . . . . . . . . . . : Yes
   Autoconfiguration Enabled . . . . : Yes

Wireless LAN adapter Local Area Connection* 2:

   Media State . . . . . . . . . . . : Media disconnected
   Connection-specific DNS Suffix  . :
   Description . . . . . . . . . . . : Microsoft Wi-Fi Direct Virtual Adapter
```

Figure 2-1: Command line showing machine properties

Step 2: Receive the floating license file from Simyog




```
#The following lines comprise the license
SERVER Khudiram 54E1AD63476E 1700
VENDOR SIMYOG
USE_SERVER
FEATURE BCI SIMYOG 1.000 30-jun-2019 2 SIGN="0038 7C21 A90A 8617
32D8 \
     0F54 7A7A ED71 C81C 3153 5D00 B81E 55D1 70BB D088 66FB 06ED
\
     79A2 EC57 9470 72A8"
FEATURE RE SIMYOG 1.000 30-jun-2019 2 SIGN="0095 73ED E294 4ED0
8D71 \
     6598 549C FDA3 CE38 7AA9 1200 688E DBB7 764B 9B69 3922 E529
\
     A48B C258 98CB F148"
```

Figure 2-2: Example floating license file

Step 3: Download the "Windows License Server" from the website http://simyog.com/download.html

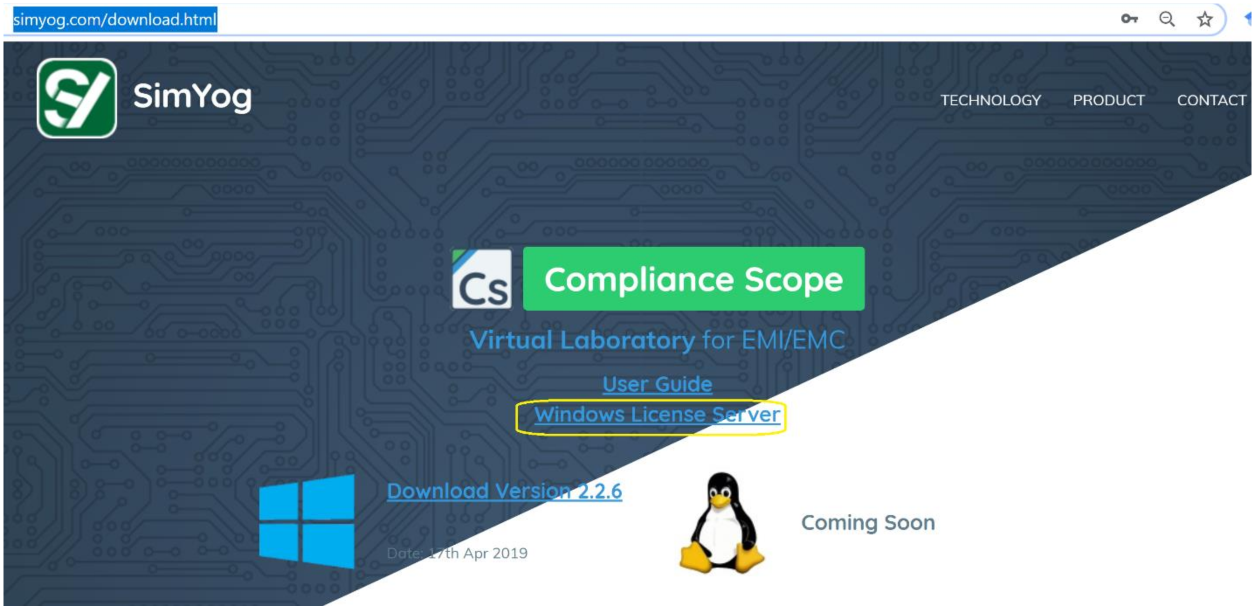


Figure 2-3: Download the license server from website

Step 4: Unzip it in a folder

Name	Date modified	Type	Size
lmgrd.exe	8/23/2017 2:43 PM	Application	1,755 KB
lmtools.exe	8/23/2017 2:41 PM	Application	1,984 KB
lmutil.exe	8/23/2017 2:43 PM	Application	1,725 KB
SIMYOG.exe	5/12/2019 4:16 PM	Application	2,260 KB
WindowsLicenseServerManual.pdf	5/13/2019 4:07 PM	Adobe Acrobat D...	172 KB

Figure 2-4: Contents of the unzipped folder

Note: It may be useful to copy the license file directly into this folder.

Step 5: Right click lmtools.exe to "Run as administrator"

Step 6: Setup the "Simyog Server"

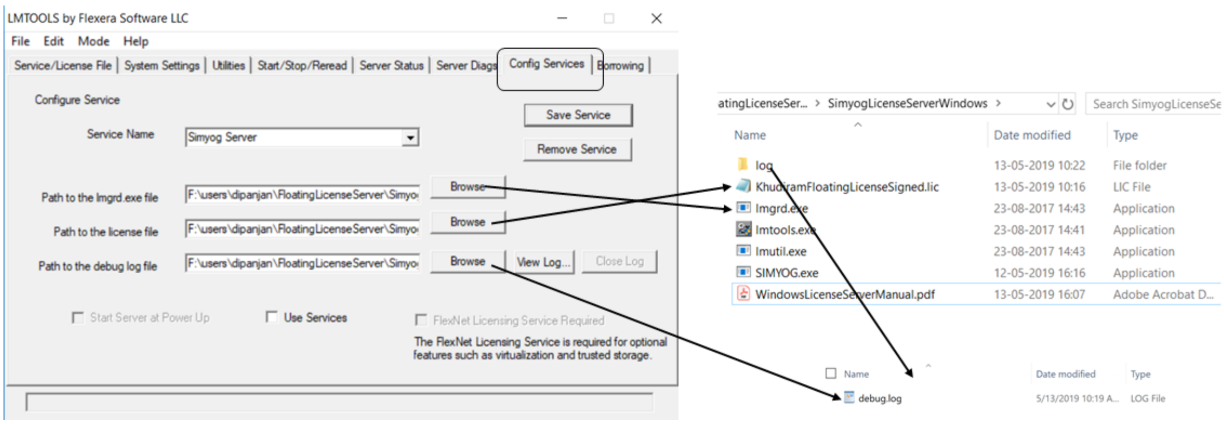


Figure 2-5: Setup the server details using lmtools

Note: please make sure to "Save Service"

Note: In case the SIMYOG.exe and lmgrd.exe are in different locations, the user needs to provide the full path name to the SIMYOG.exe in the license file

```
#The following lines comprise the license
SERVER Khudiram 54E1AD63476E 1700
VENDOR SIMYOG D:\temp\SIMYOG.exe
USE_SERVER
FEATURE BCI SIMYOG 1.000 30-jun-2019 2 SIGN="0038 7C21 A90A 8617
32D8 \
     0F54 7A7A ED71 C81C 3153 5D00 B81E 55D1 70BB D088 66FB 06ED
\
     79A2 EC57 9470 72A8"
FEATURE RE SIMYOG 1.000 30-jun-2019 2 SIGN="0095 73ED E294 4ED0
8D71 \
     6598 549C FDA3 CE38 7AA9 1200 688E DBB7 764B 9B69 3922 E529
\
     A48B C258 98CB F148"
```

Figure 2-6: License file with the location of the SIMYOG daemon

Step 7: Start the server

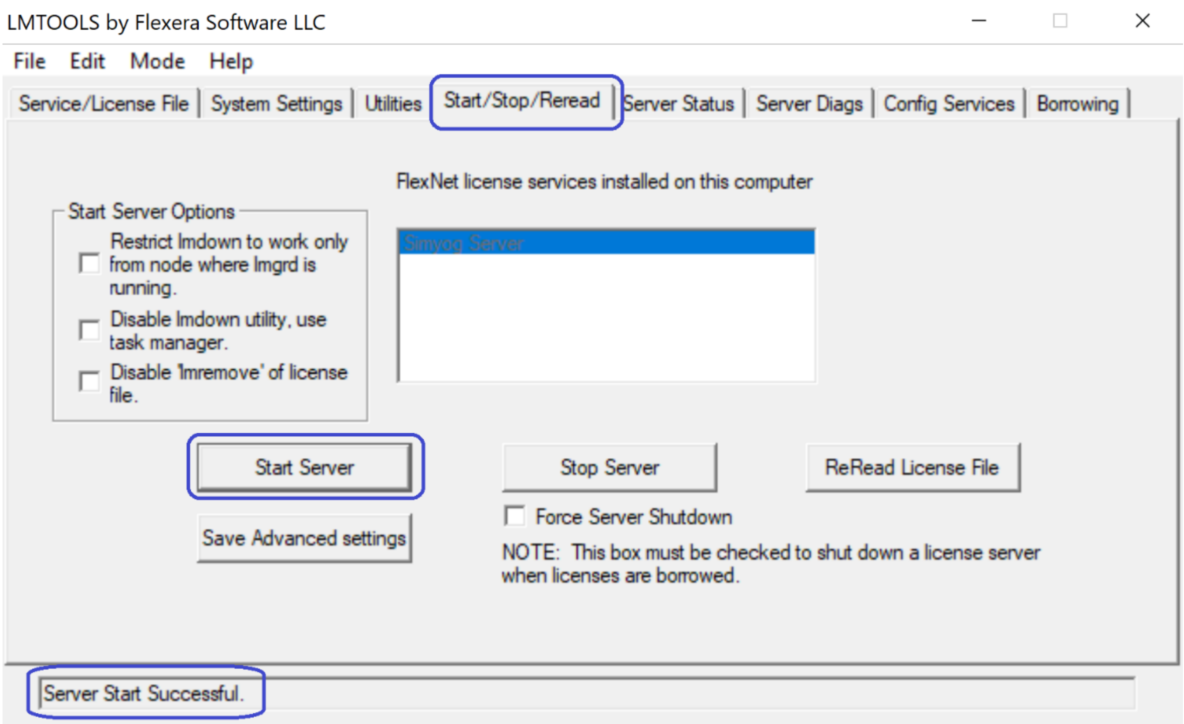


Figure 2-7: Starting the server from lmtools

Step 8: Monitor the server licenses

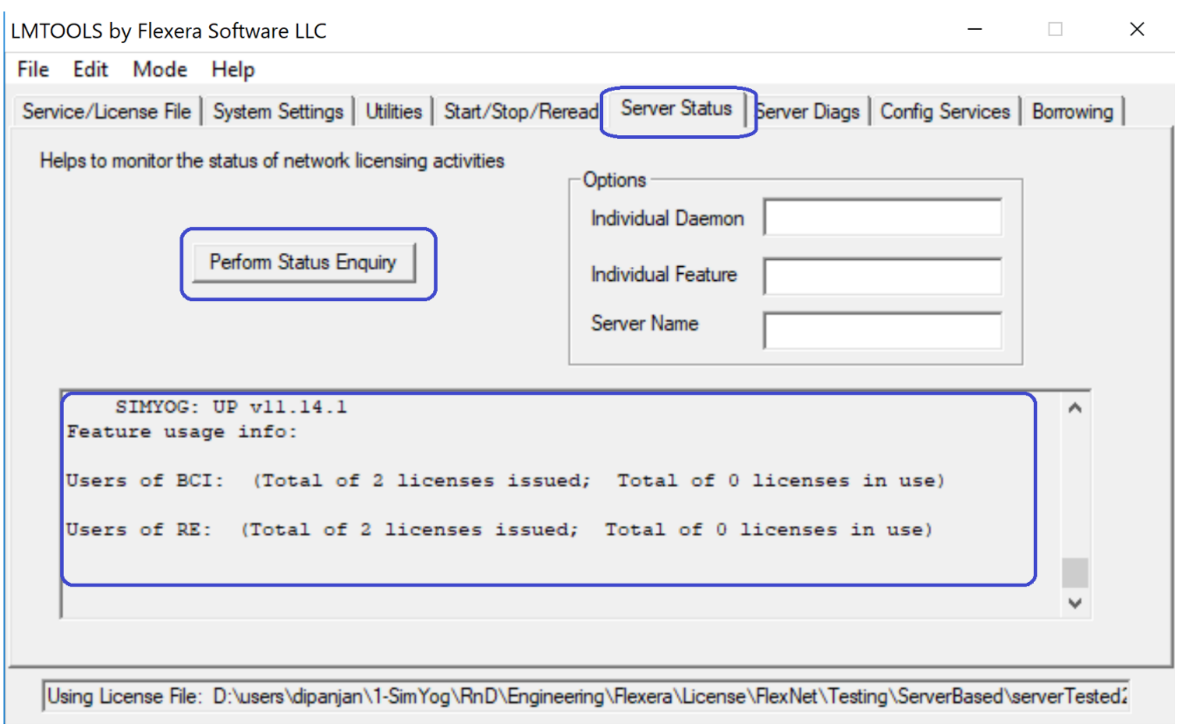


Figure 2-8: Monitor the server using lmtools

2.3.1.2 Client Side

Step 1: Install CompScope

Step 2: Start CompScope as described in 2.4 Installation Steps

On the license screen select "" and provide the port@ipaddress (here 1700@192.168.0.111)

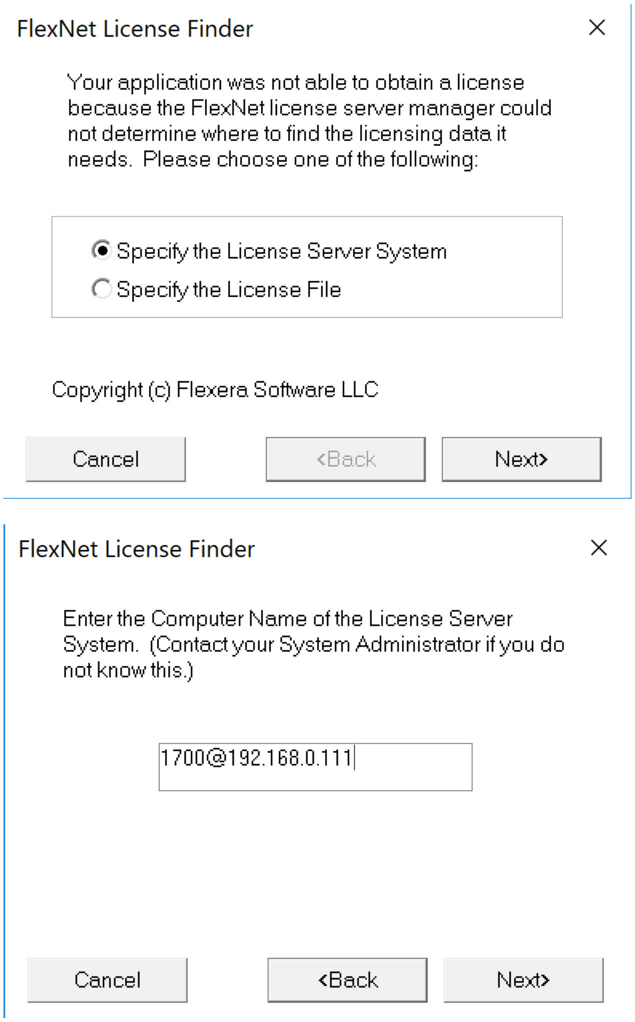


Figure 2-9: Selecting the license server system

2.3.1.3 Connectivity issues

A common problem with the license server may arise from the windows firewall blocking the client request to the server from checking out or releasing licenses.

One way to prevent this problem is to make exceptions for lmgrd.exe and SIMYOG.exe in the firewall settings.

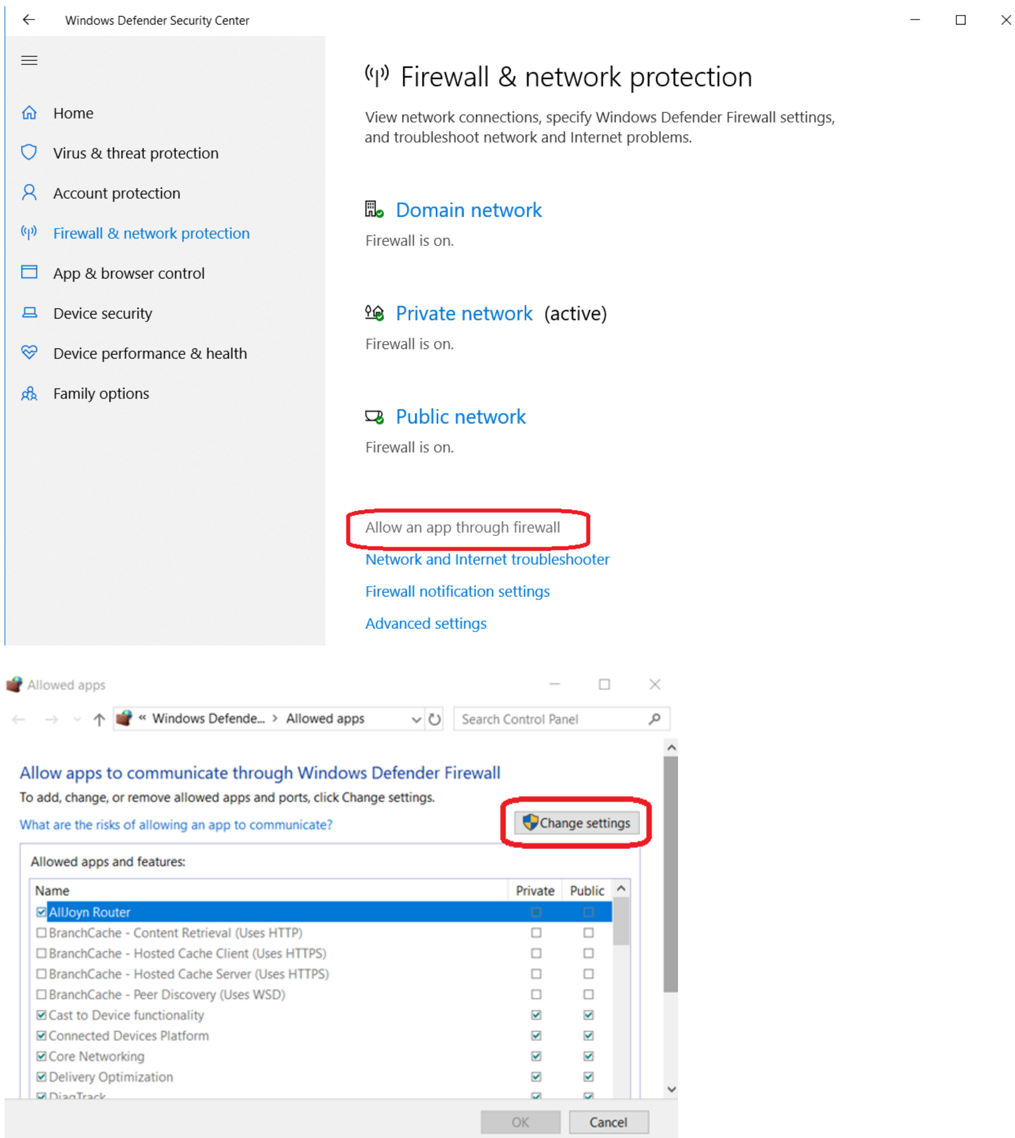


Figure 2-10: Change settings to enable adding new programs to exception list

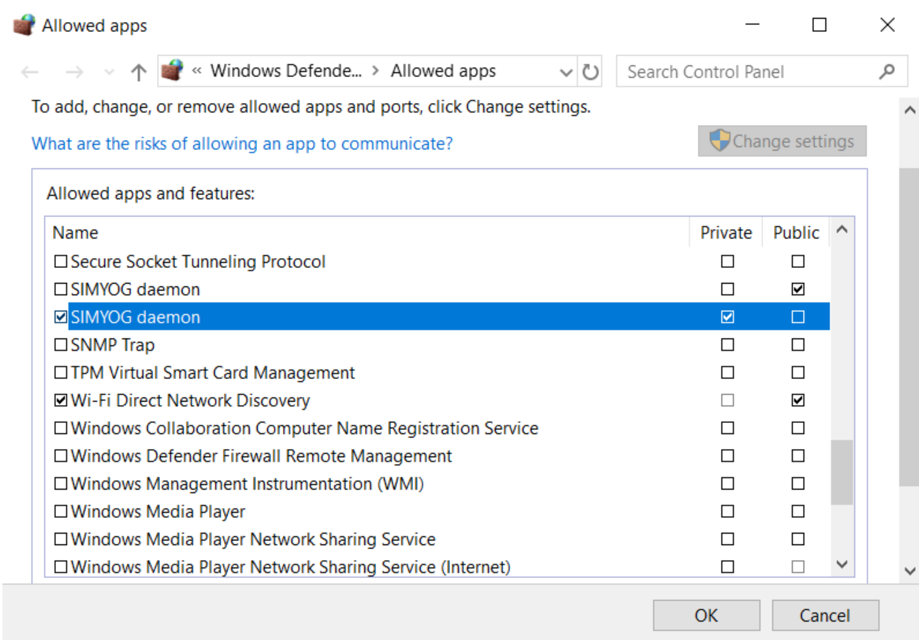


Figure 2-11: Adding SIMYOG.exe from the unzipped folder by browsing

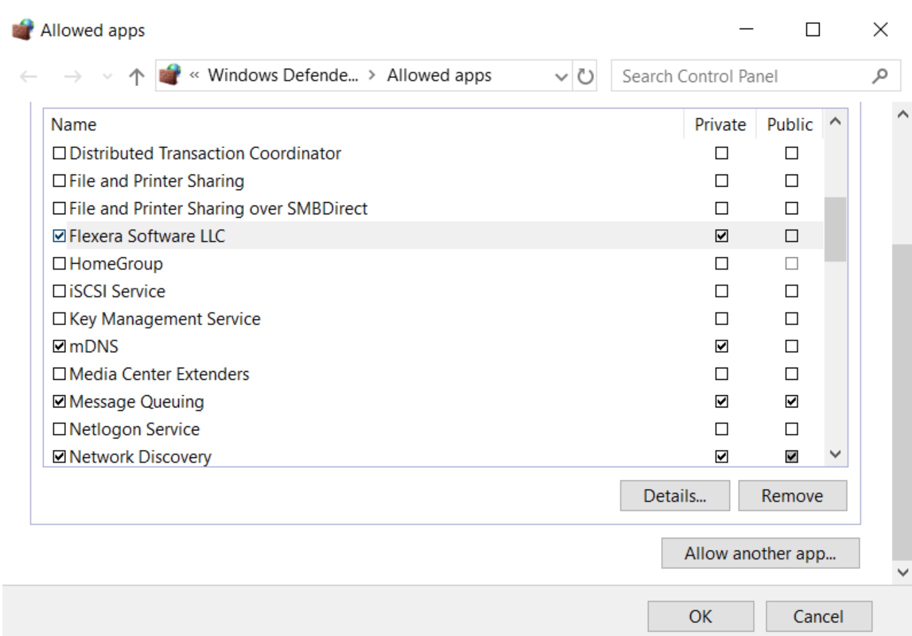


Figure 2-12: Adding lmgrd.exe from the unzipped folder by browsing

2.4 Installation Steps

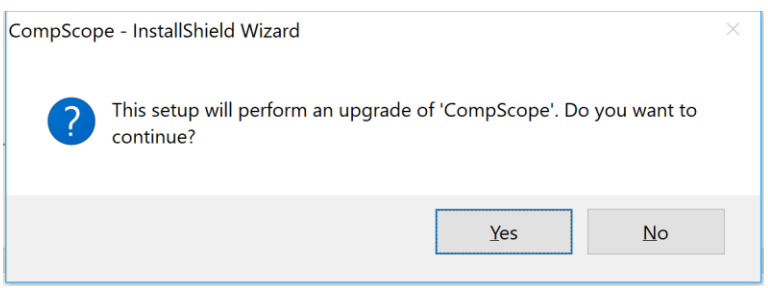
Step 1: Go to the link http://www.simyog.com/download.html to download the Compliance Scope software.

Figure 2-13: CompScope Download Page

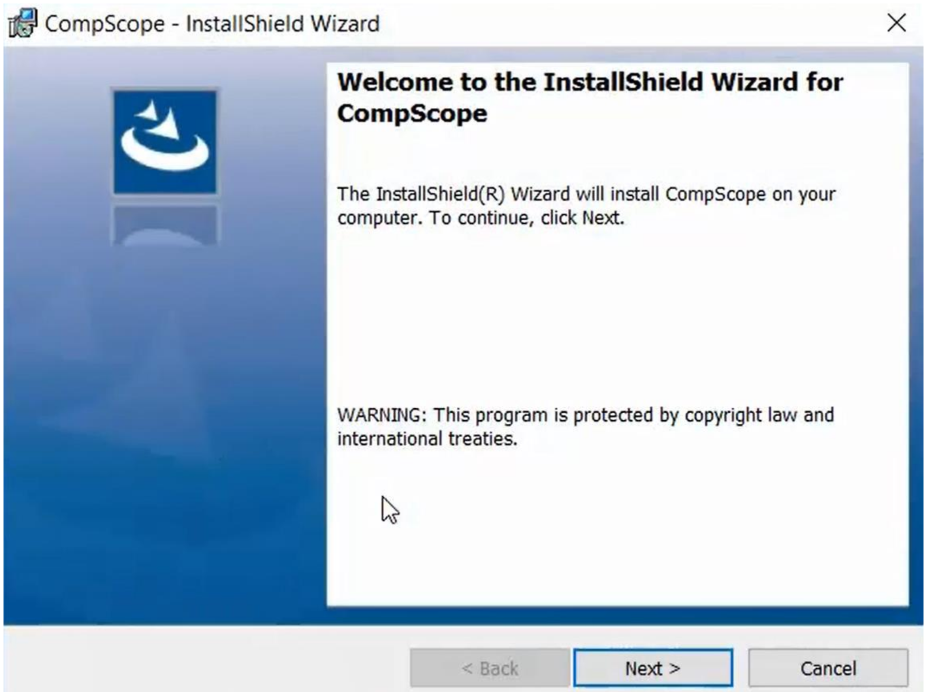
Step 2: Click on the desired version and fill up the appropriate username and password to start the download of CompScope_X.Y.Z_setup.exe.

Step 3: Double click on CompScope_X.Y.Z_setup.exe.

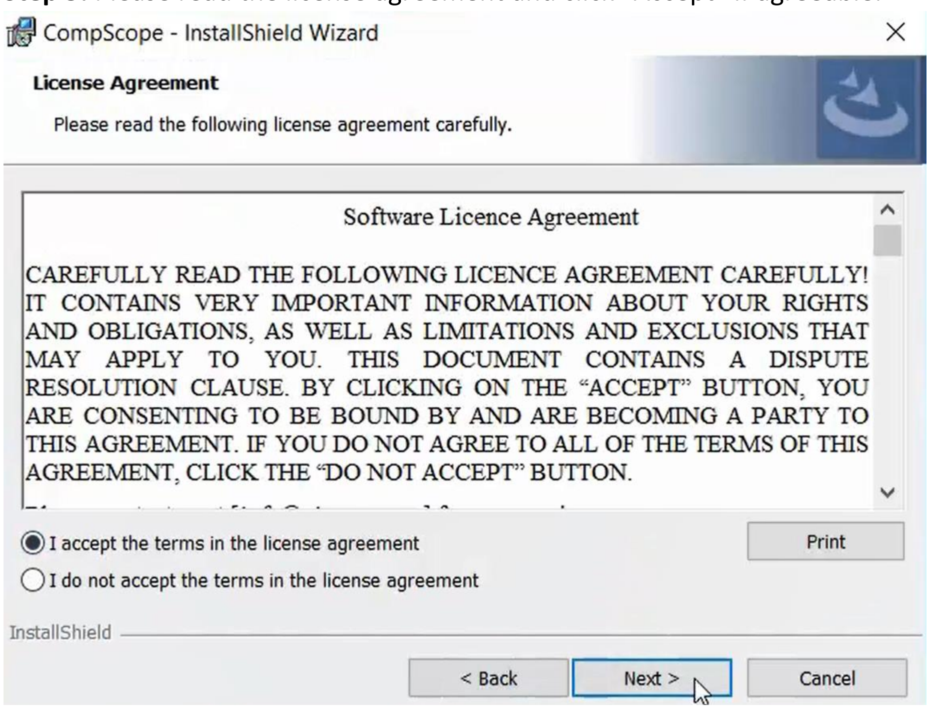
(If you are installing over existing older version of CompScope, the following widget appears. Click "Yes" to continue.)



Step 4: Click "Next" to continue.



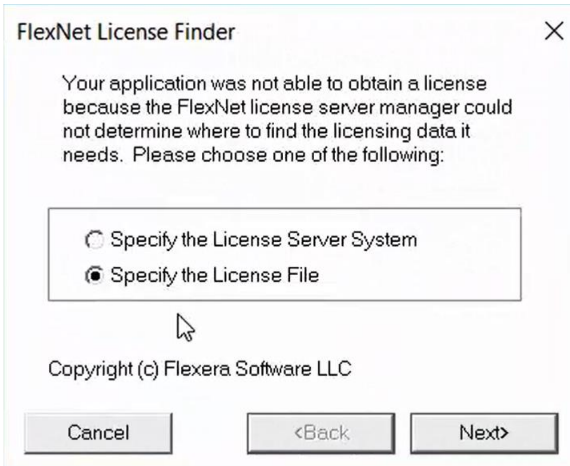
Step 5: Please read the license agreement and click “Accept” if agreeable.



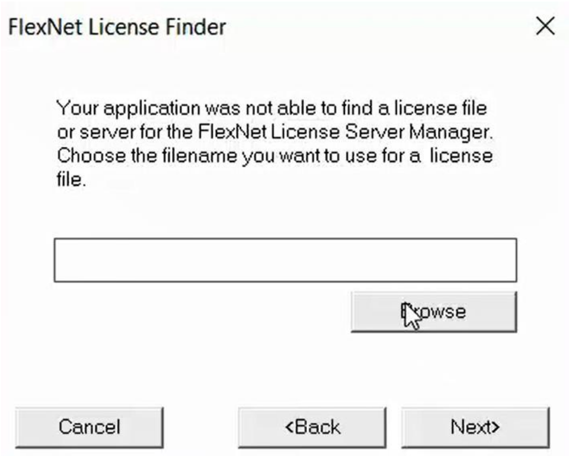
Step 6: Follow the instructions and continue as prompted.

Step 7: Check the "Launch CompScope" box and click "finish" to complete the installation. CompScope opens up. Click on "BCI" to proceed.

Step 8: FlexNet® License Finder window pops up. Select "Specify the License File" and click "Next".



Step 9: Browse to the location of the license file and click "Next"



Step 10: Click "Finish" to complete the installation and licensing. Compliance Scope is now ready for use.

2.5 Useful Resources

In the course of following this user guide, the reader may also benefit from the following resources:

(a) Installation movie: http://simyog.com/resources/CSinstallation.mp4

(b) Explanation of the different GUI widgets: http://simyog.com/resources/GettingStarted.mp4

(c) Starting resources for (b): http://simyog.com/resources/BCIresources.zip

(d) Training labs including solve and results: Chapter 11: Example CompScope Experiments

3 Chapter 3: Getting Started

This chapter provides a description of the opening two screens of CompScope: (i) the experiment catalog screen and (ii) experiment screen.

3.1 Experiment-Catalog Screen:

On clicking the installed Compliance Scope executable, the experiment-catalog window opens as shown in Figure 3-1:

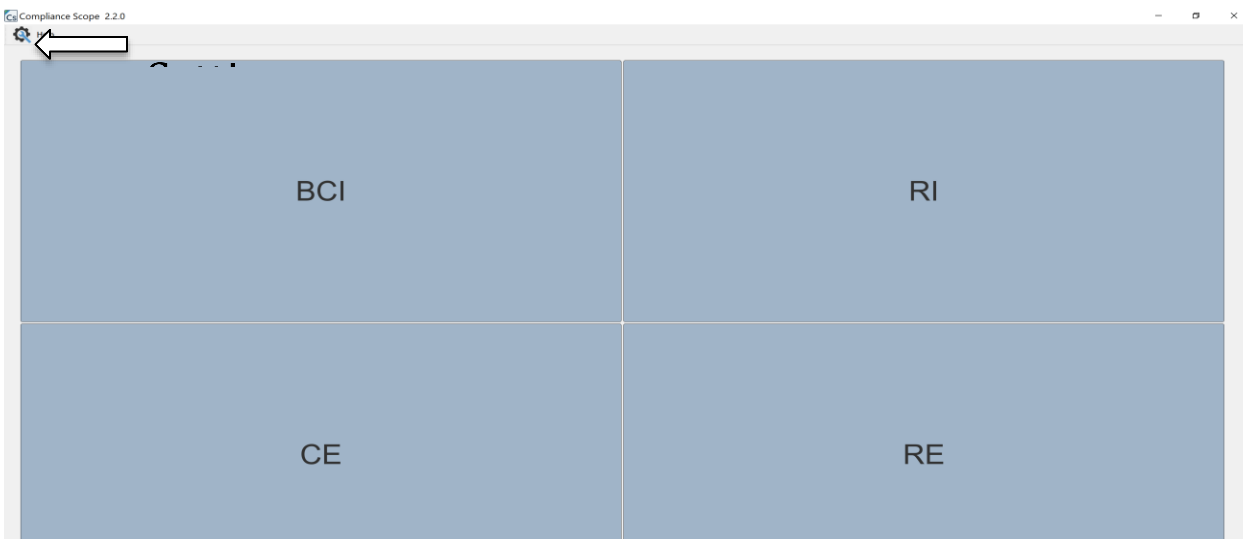
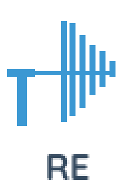


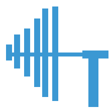
Figure 3-1: CompScope Experiment Catalog

Each tile represents a type of experiment. The 6 experiments that will be supported by CompScope are:

Bulk Current Injection: The Bulk Current Injection (BCI) method is mainly used for RF immunity testing of automotive components and ICs in electronics industry. In this test method, current is injected on one or more pins simultaneously to test for malfunctions.

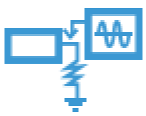


Radiated Emissions: Radiated emissions testing involves measuring the electromagnetic field strength of the emissions that are unintentionally generated by the device. Emissions are inherent to the switching voltages and currents in the device.



RI

Radiated Immunity: Radiated immunity test is to measure the immunity of device when subject to prescribed radiated electromagnetic field.



CE

Conducted Emissions: Conducted emission is performed to evaluate the radio frequency noise present in the physical wiring or traces of an electrical system due to any switching or harmonic resonances within an equipment.

Note: Up to the current version of CompScope (v 2.2.0), only BCI and RE tests are supported. The other tests will be released shortly.

On clicking on any of the tiles, the corresponding Experiment Main Screen opens as shown in Figure 3-4. Only 1 experiment instance can be opened at a time.

3.1.1 Settings Window

On clicking the "Settings Button" from the Experiment-Catalog Screen as shown in Figure 3-1, the settings window will appear as shown below in Figure 3-2.

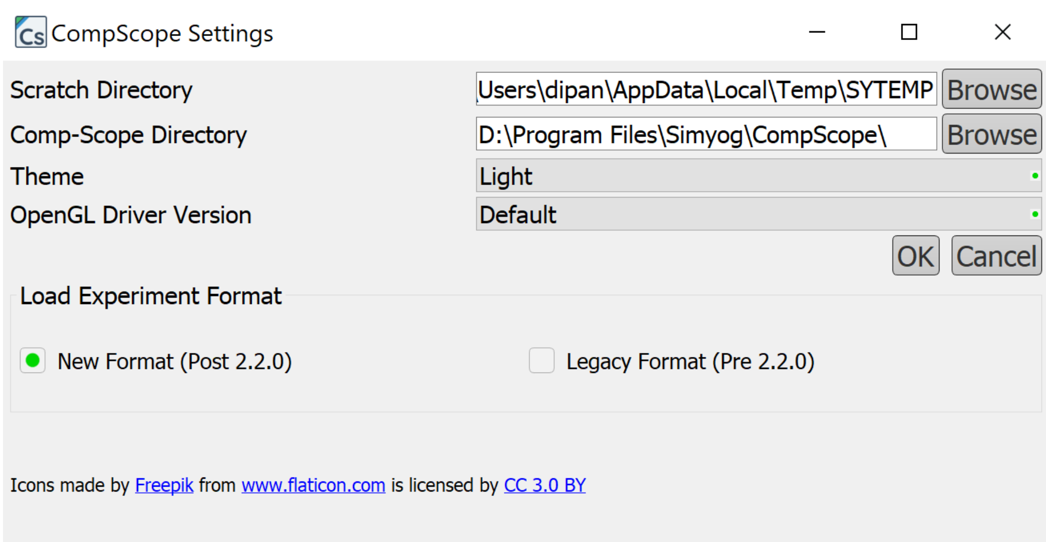


Figure 3-2: CompScope Settings Window

“Scratch Directory” refers to a location where CompScope will generate a folder where run-time files are printed. “CompScope Directory” refers to the path where the CompScope.exe has been installed. “Theme” refers to the color scheme used in the GUI and can be either Light OR Dark. “OpenGL Driver Version” refers to the version of OpenGL that CompScope will use for GUI rendering. It can be selected from among “Default” (recommended), “OpenGL2” and “OpenGL”.

If loading an experiment saved by a lower version than 2.2.0, the “Legacy Format” checkbox needs to be checked.

These variables are populated automatically during the installation of CompScope. Usually in the normal mode of operations, the user will not require to change/modify any of these variables.

Note: For advanced readers, these settings variables are stored and cached from registry. You can see them using regedit under HKEY_CURRENT_USER -> Software -> Simyog as shown in Figure 3-3*. These three variables are derived from the CompScope settings and used inside the CompScope software.*

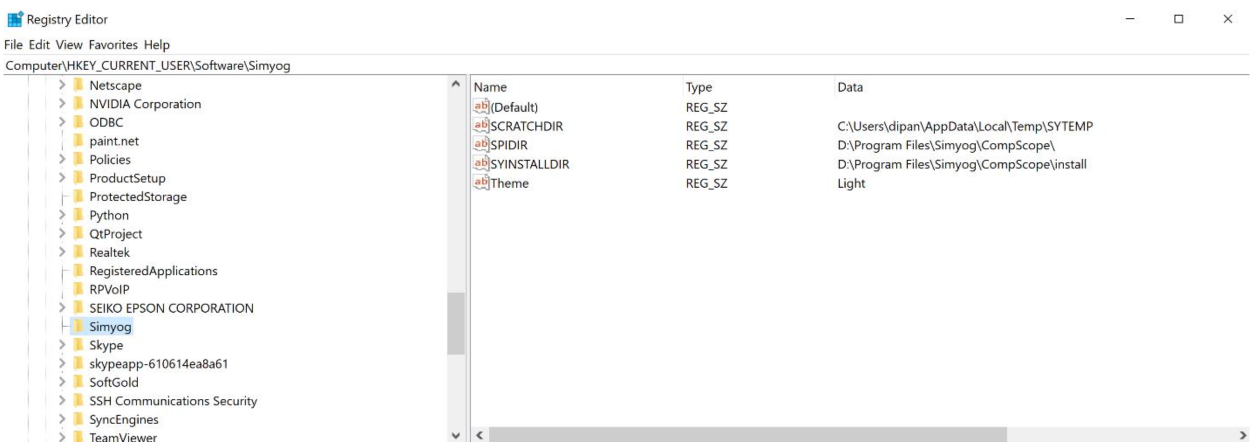


Figure 3-3: Registry items for CompScope Settings

Note: It is advisable to clean, i.e. delete contents of, the SCRATCHDIR regularly to prevent unnecessary data storage

3.2 Experiment Main Screen

On clicking any of the experiment-catalog tiles, the corresponding experiment is setup automatically for the user. For example, on clicking the “BCI” tile, the BCI experiment main window appears as shown in Figure 3-4 and the corresponding for “RE” is shown in Figure 3-5: RE Experiment Main Screen.

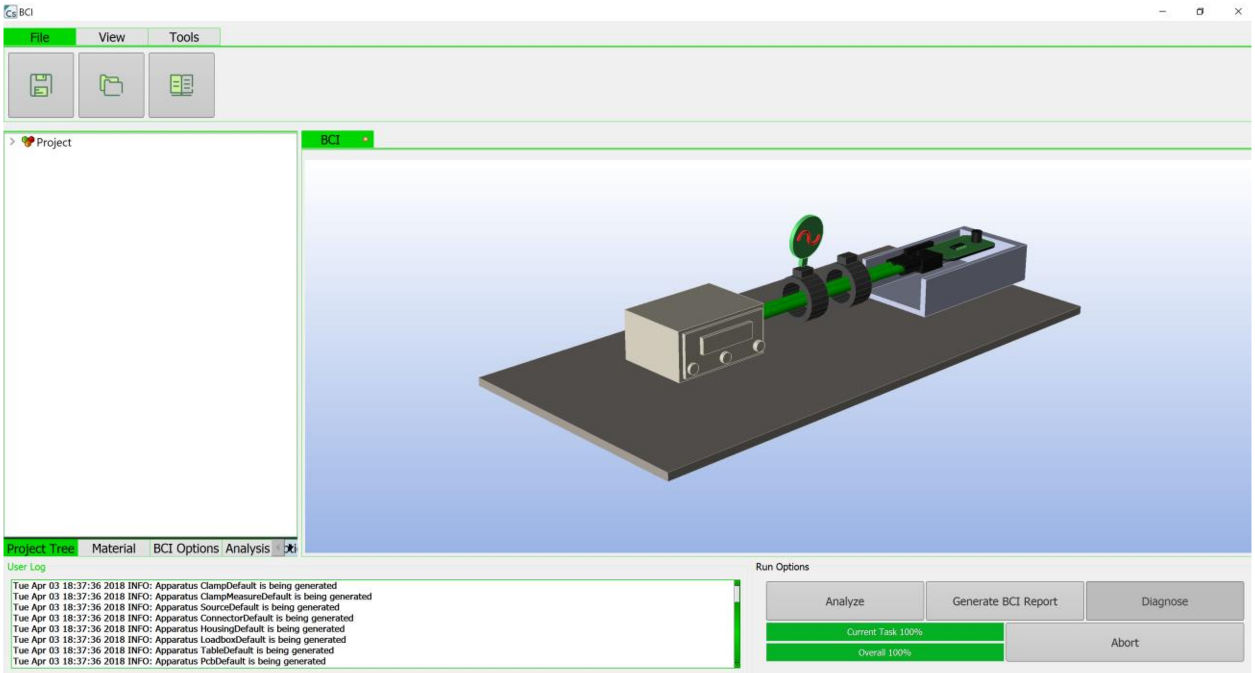


Figure 3-4: BCI Experiment Main Screen

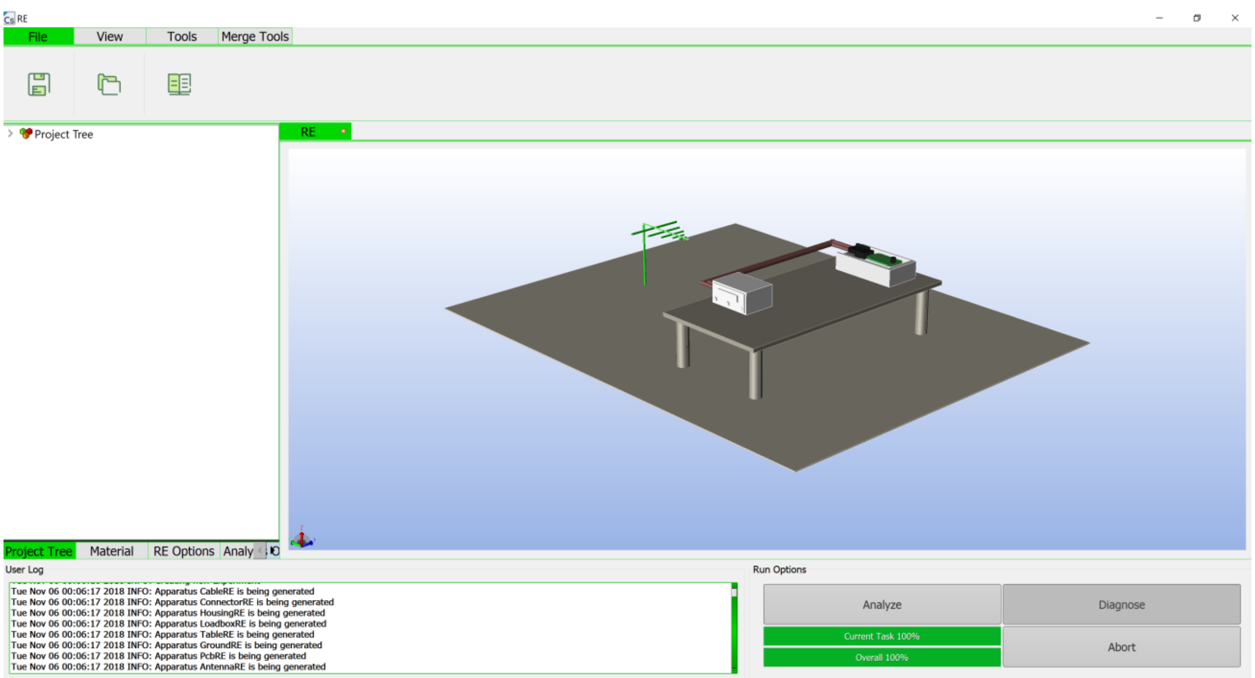


Figure 3-5: RE Experiment Main Screen

From a functionality perspective, the experiment main screen can be divided into several segments: (i) Utility Buttons (ii) Log messages (iii) Run screen and (iv) Main Experiment (BCI/RE) Window OR Individual Apparatus Window, as shown in Figure 3-6.

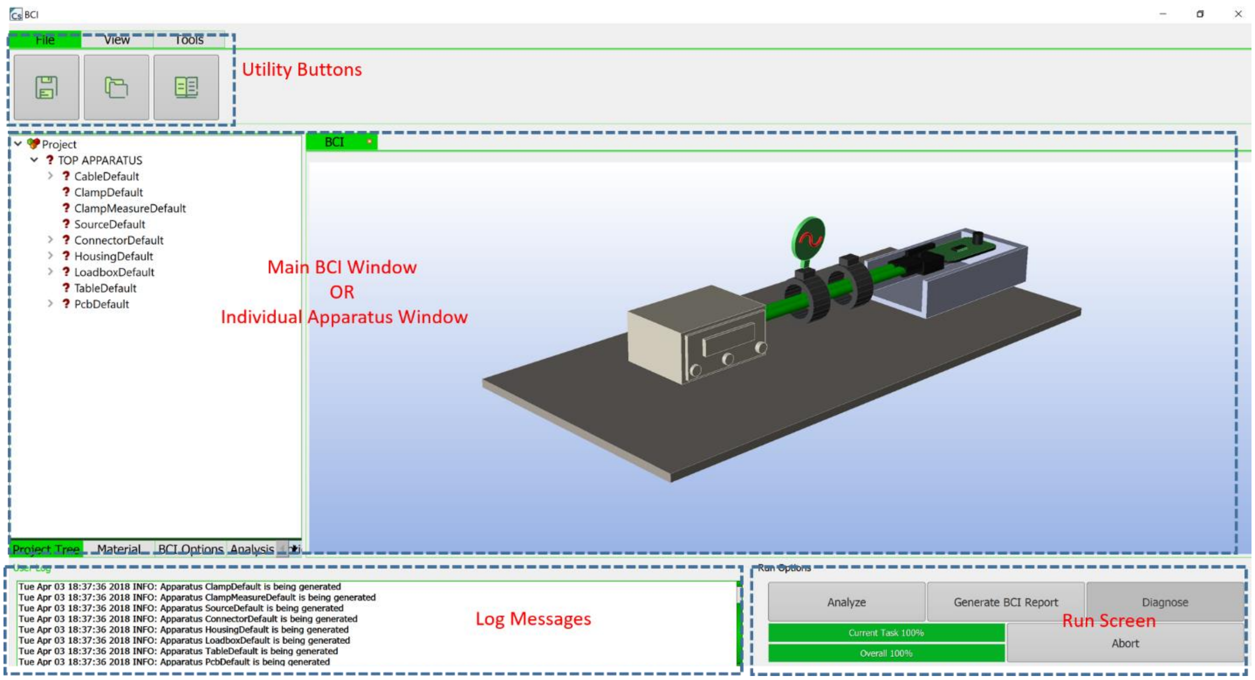


Figure 3-6: Functional blocks of the Experiment Main Screen

3.2.1 BCI/RE Main Window OR Individual Apparatus Window

This space is common to the BCI/RE main window (discussed in Chapter 4: Experiment (BCI/RE) Main Window) and the Individual Apparatus Windows (discussed in Chapter 6: Individual Apparatus Window). The switching between the windows is accomplished by clicking on the appropriate tab as shown in Figure 3-7.

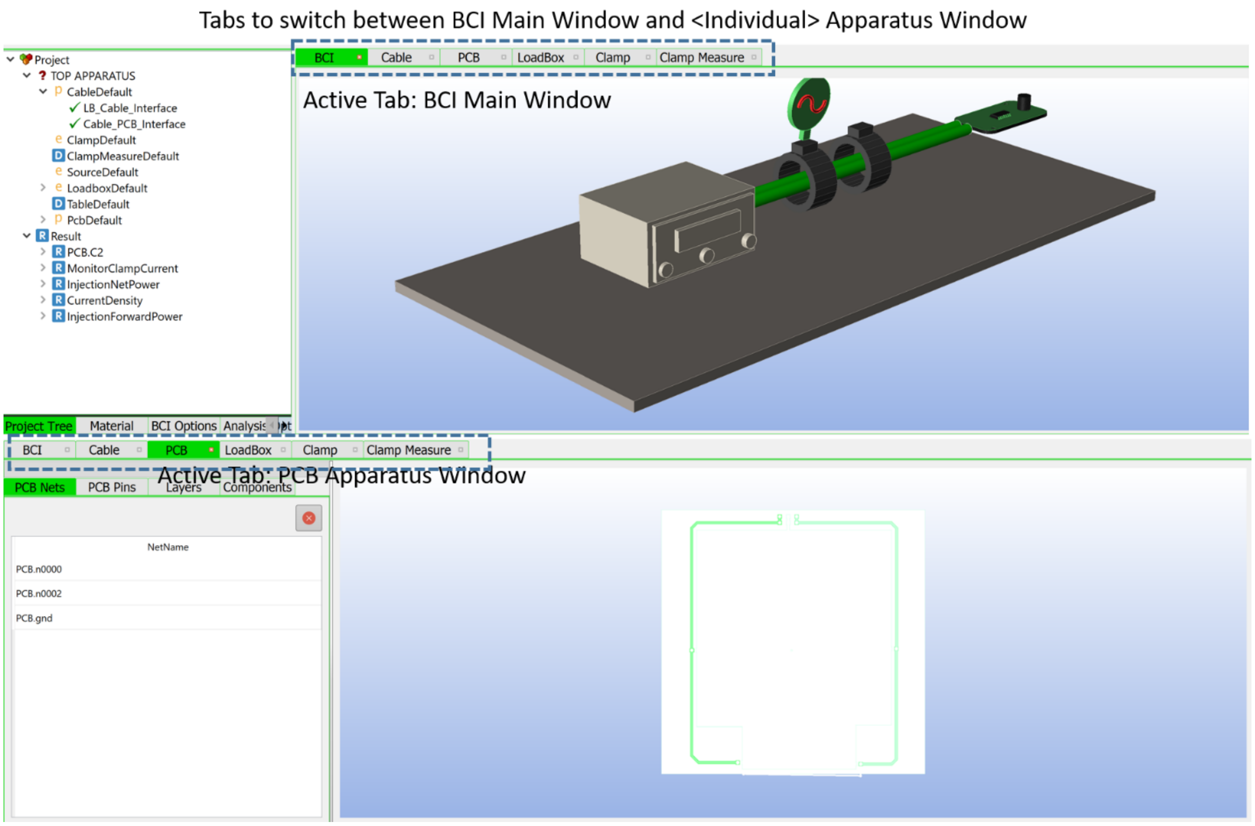


Figure 3-7: Tabbed BCI Main Window and Individual Apparatus Window

3.2.2 Utility Buttons

The “Utility Buttons” are common to “Main Apparatus Window” or any of the “<Individual> Apparatus Window”.

3.2.2.1 File Menu

The utilities in File Menu consists of “Save”, “Load” and “Library” buttons.

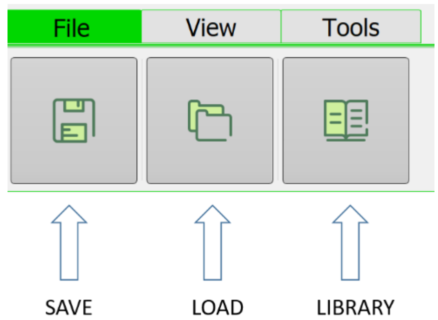


Figure 3-8: Utility Buttons

For the BCI/RE main window or the Individual Apparatus Windows the File Menu options have different functionalities as shown in Table 2.

Table 2: File menu options

Window	**Experiment Type**	**Save**	**Load**	**Library**
Experiment Main Window	BCI/RE	Experiment file (.bci/.re)	Experiment file (.bci/.re)	NA
Cable Apparatus Window	BCI/RE	NA	NA	NA
Clamp Apparatus Window	BCI	s3p file	s3p file	Clamp-F140 (Recommended)
Measurement Clamp Apparatus Window	BCI	NA	NA	NA
Source Apparatus Window	BCI	NA	NA	NA
Loadbox Apparatus Window	BCI/RE	sp or s*p	sp or s*p	NA
Connector Apparatus Window	BCI/RE	3D file (.sat, .igs, .step)	3D file (.sat, .igs, .step)	NA
Housing Apparatus Window	BCI/RE	3D file (.sat, .igs, .step)	3D file (.sat, .igs, .step)	NA
Table Apparatus Window	BCI/RE	NA	NA	NA

Antenna Apparatus Window	RE	NA	NA	NA
PCB Apparatus Window	BCI/RE	3D file (.sat, .igs, .step)	ODB++	NA

3.2.2.2 View Menu

The view buttons include the following options:

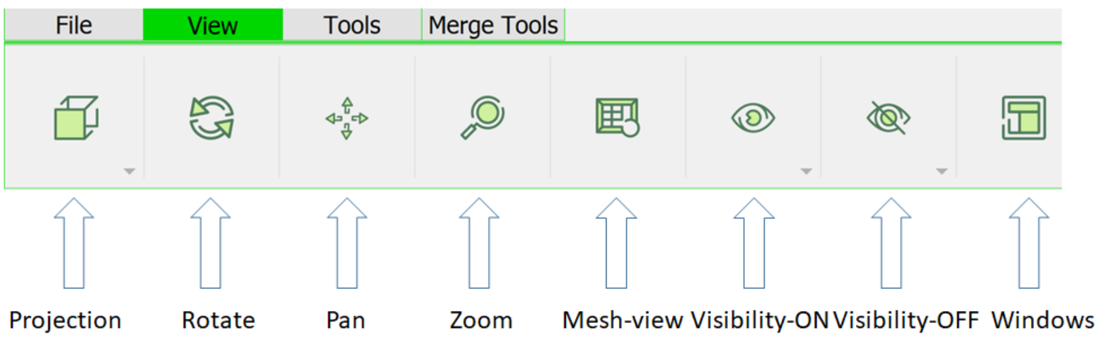


Figure 3-9: View buttons

Projection: Geometry visualization from different angles – top, bottom, side etc.

The demonstration link is: http://simyog.com/resources/manual_animations/Projections.mp4

Rotate: To rotate the geometry by an angle

The demonstration link is: http://simyog.com/resources/manual_animations/Rotate.mp4

Pan: To move the geometry by dragging

The demonstration link is: http://simyog.com/resources/manual_animations/Pan.mp4

Zoom: To zoom in to the geometry by box selection

The demonstration link is: http://simyog.com/resources/manual_animations/Zoom.mp4

Mesh-view: To view the mesh on the geometry

The demonstration link is: http://simyog.com/resources/manual_animations/MeshView.mp4

Visibility-ON or Visibility-OFF: Turn the visibility of dielectrics, pins and components

The demonstration link is: http://simyog.com/resources/manual_animations/Visibility.mp4

Window: To automatically go to default window tab system

The demonstration link is: http://simyog.com/resources/manual_animations/Window.mp4

3.2.2.3 Tools Menu

The tools menu includes the following options:

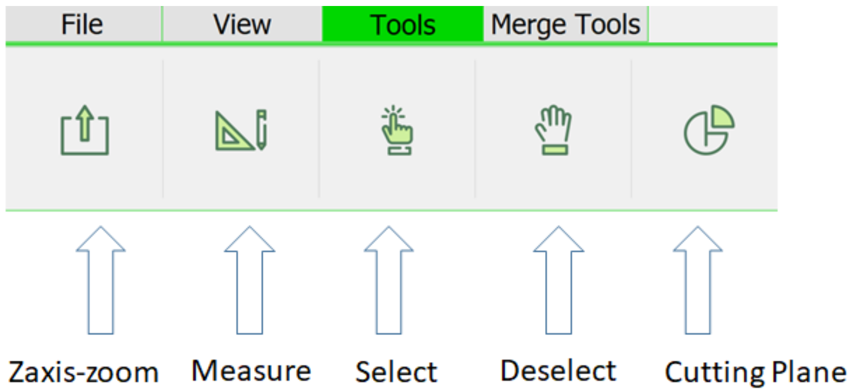


Figure 3-10: Tools menu buttons

Zaxis-zoom: Zooming the z direction to study the layers of PCB in particular

The demonstration link is: http://simyog.com/resources/manual_animations/ZaxisZoom.mp4

Measure: Measuring distance between points on geometry

The demonstration link is: http://simyog.com/resources/manual_animations/Measure.mp4

Select: Selecting nets, pins, components etc.

The demonstration link is: http://simyog.com/resources/manual_animations/Selection.mp4

De-Select: De-Selecting nets, pins, components etc.

Cutting Plane: Cutting objects for internal visibility – particularly useful in merging operations

3.2.2.4 Plot Options

The plot options only appear if results are present in the BCI/RE Main Window. It consists of the following options:

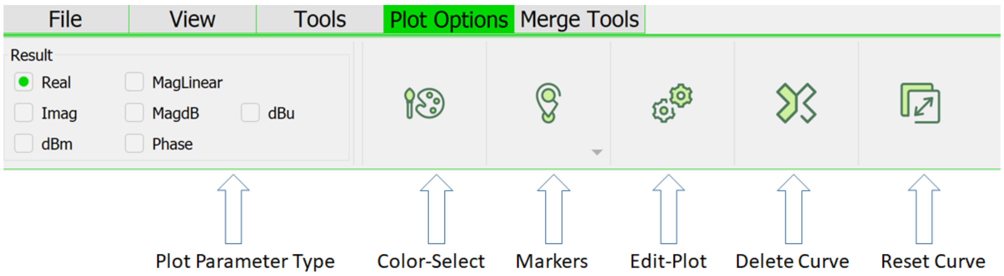


Figure 3-10: Plot Options Menu

Details of the plot options can be found in Chapter 10: Results.

3.2.3 Log Message Window

Log window shows runtime Error and Warnings as well as other information messages generated by CompScope. This Log Window is common across the BCI/RE main window and Individual Apparatus Windows and user should refer to log in case any operation fails in Apparatus Windows.

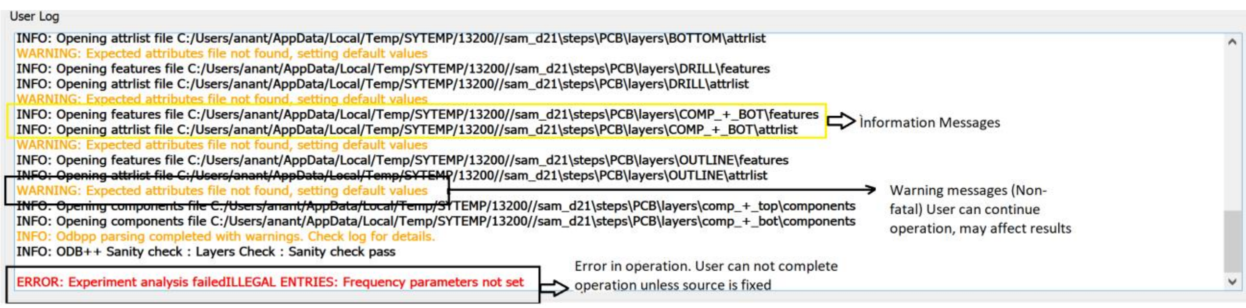


Figure 3-11: Example log window

3.2.4 Run Window

The Run window, as shown in Figure 3-12, is used for triggering analysis:

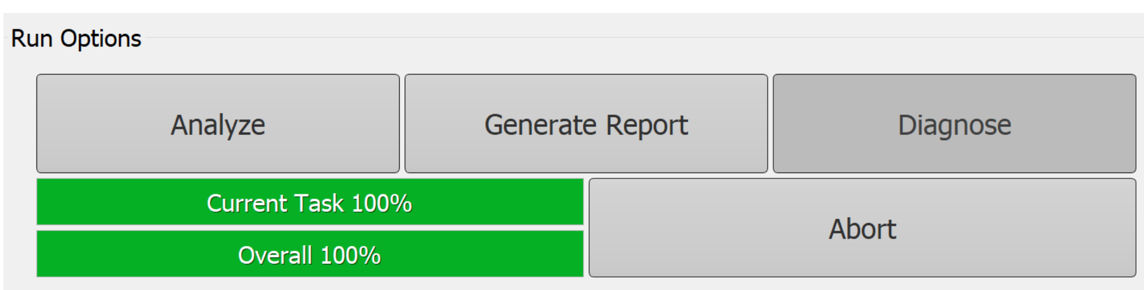


Figure 3-12: Run Window (BCI)

3.2.4.1 Analyze:

This triggers the solution of the setup experiment

While, the “analyze” operation runs, log messages will appear like:

```
INFO: 3D Full-wave Iterative: Solving Frequency (Index 0): Value: 1.000000 MHz Cores: 8 ...
INFO: 3D Full-wave Iterative: Solving Frequency (Index 1): Value: 22.000000 MHz Cores: 8 ...
INFO: 3D Full-wave Iterative: Solving Frequency (Index 2): Value: 43.000000 MHz Cores: 8 ...
```

3.2.4.2 Generate Report

This should be run after a successful "Analyze" step to generate BCI or RE report. More details about "Generate Report" can be found in 10.4 Generate Report.

3.2.4.3 Diagnosis

This is a feature that is not currently supported, but will be enabled shortly

3.2.4.4 Progress bar

This consists of 2 levels of progress – the one at the bottom is the global progress bar and the one on the top corresponds to that of individual tasks constituting the global progress.

3.2.4.5 Abort button

This is used to interrupt the run in the middle. The actual abort may take a few seconds after being pressed, since it has to kill all running threads. Once the abort is completed, the log message, as shown below will appear:

```
ERROR: Experiment analysis failedABORTED BY USER: Aborted by user
```

4 Chapter 4: Experiment (BCI/RE) Main Window

The BCI/RE Main Window comprises of 2 segments as shown below:

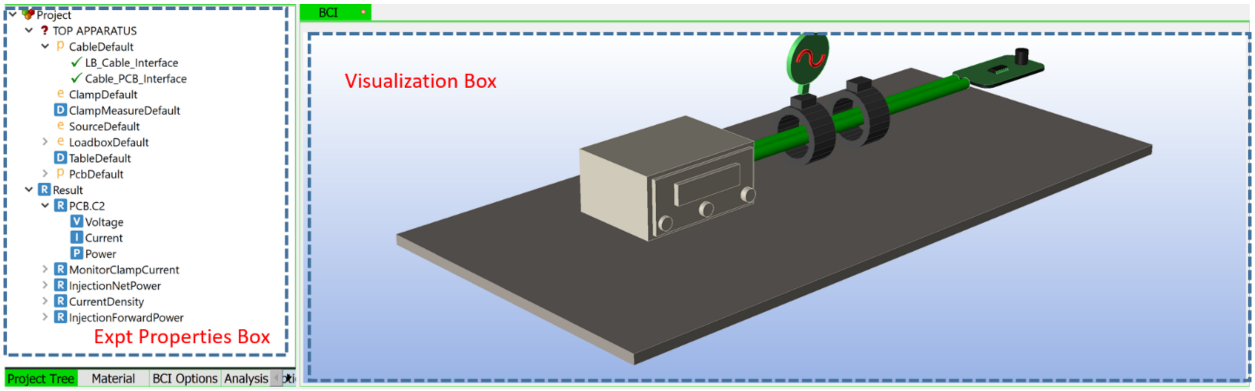


Figure 4-1: BCI Main Window components

(a) Experiment Properties Box containing

Project Tree (explained in 0

- Project Tree)
- Materials Window (explained in Chapter 5: Materials)
- BCI OR RE options (explained in 4.3 BCI options and 4.4 RE options)
- Analysis Options (explained in 4.5 Analysis options)
- Mesh Options (explained in 4.6 Mesher options)

(b) Visualization box

4.1 Experiment load and save:

Load: To load a saved experiment, click on the "Load Button" and browse to a saved .bci OR .re file as shown in Figure 4-2.

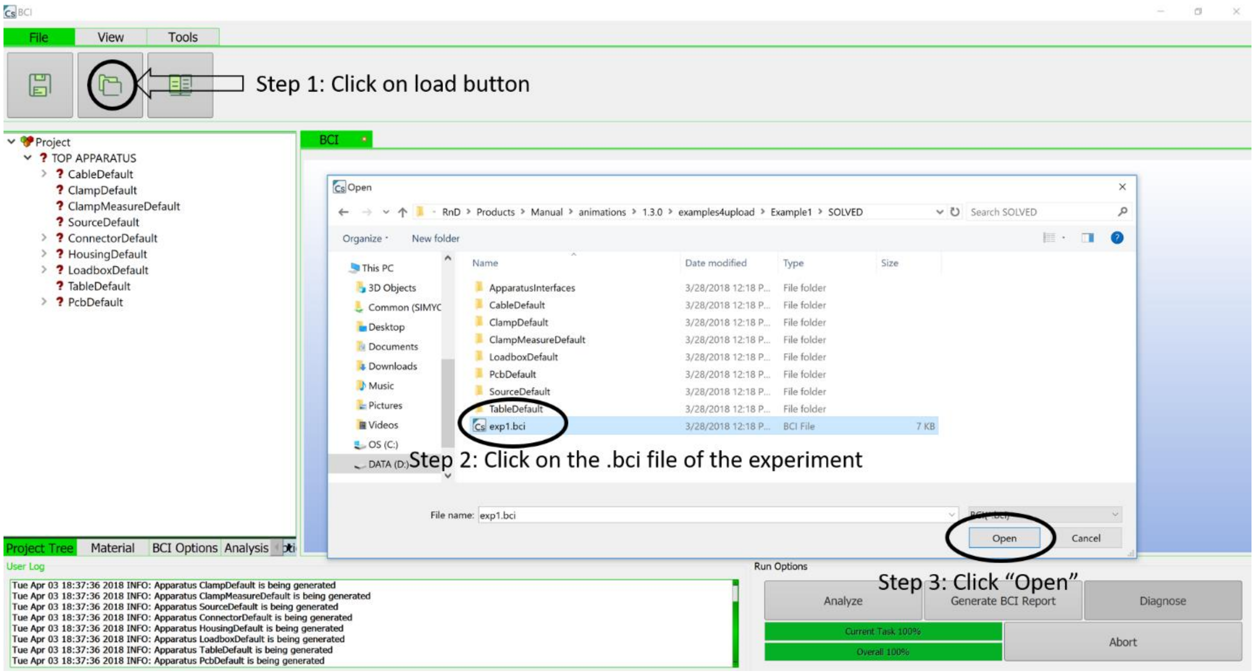


Figure 4-2: Loading an existing experiment

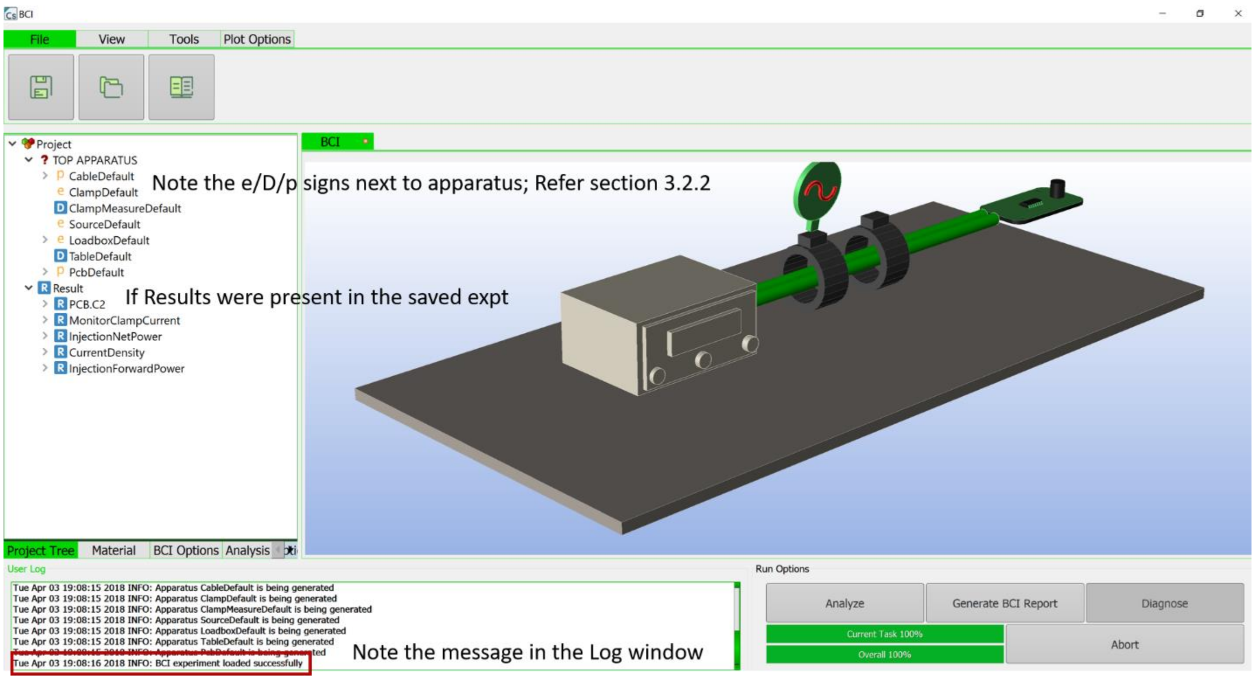


Figure 4-3: Loaded experiment

Save: To save an experiment click the “save button”, browse to the desired folder and enter an experiment name as shown in Figure 4-5.

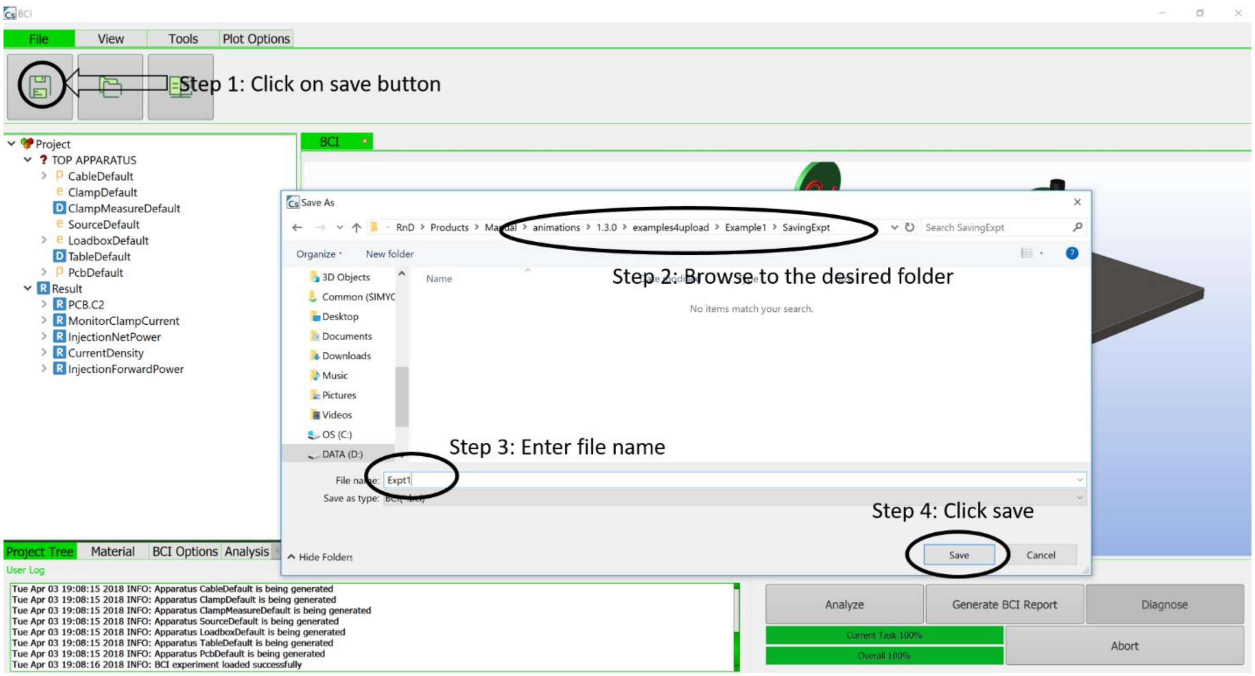


Figure 4-4: Saving an experiment

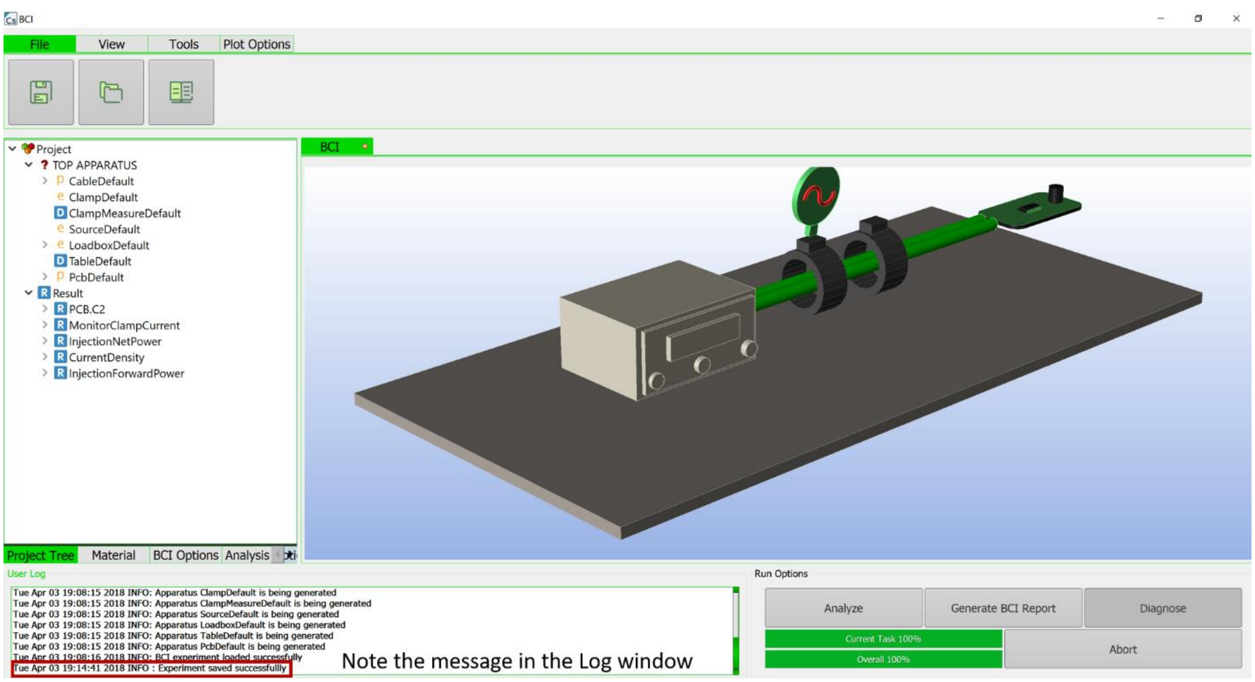


Figure 4-5: After saving the experiment

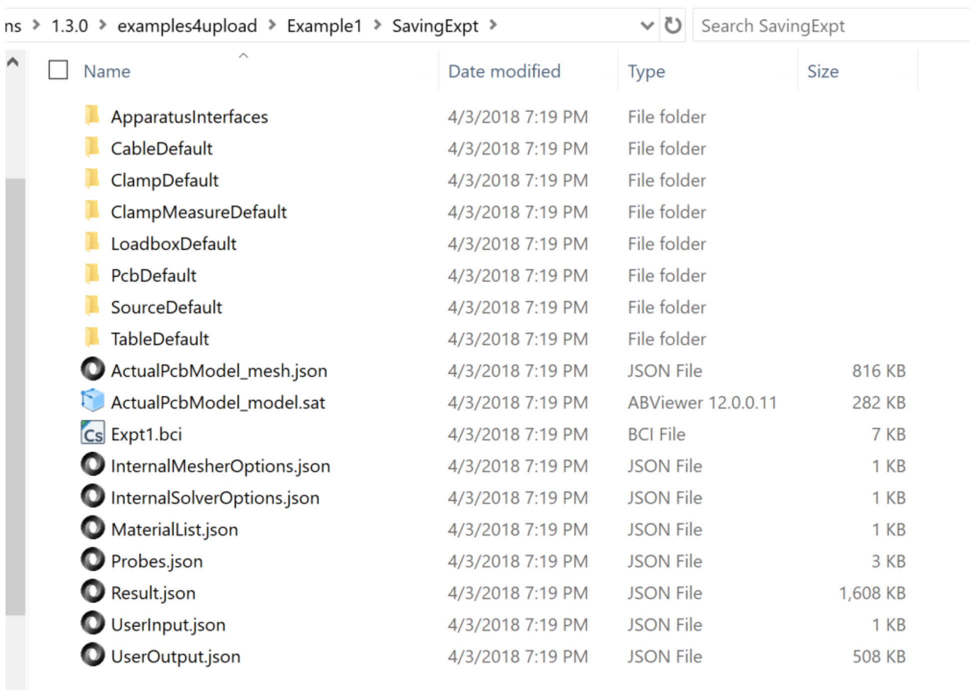


Figure 4-6: Saved experiment structure

Note: It is highly recommended that the experiment be saved in an empty folder (in this case, the "SavingExpt" folder. This way all its components are clearly visible.

Library: Load experiment from library.

Note: The load experiment from library option is disabled in the current version (v 1.3.0).

4.2 Project Tree

4.2.1 Apparatuses

Each apparatus can be traced to its visual model in the “Visualization screen”. The mapping is shown for a BCI experiment in Figure 4-7 and an RE experiment in Figure 4-8: RE experiment apparatuses.

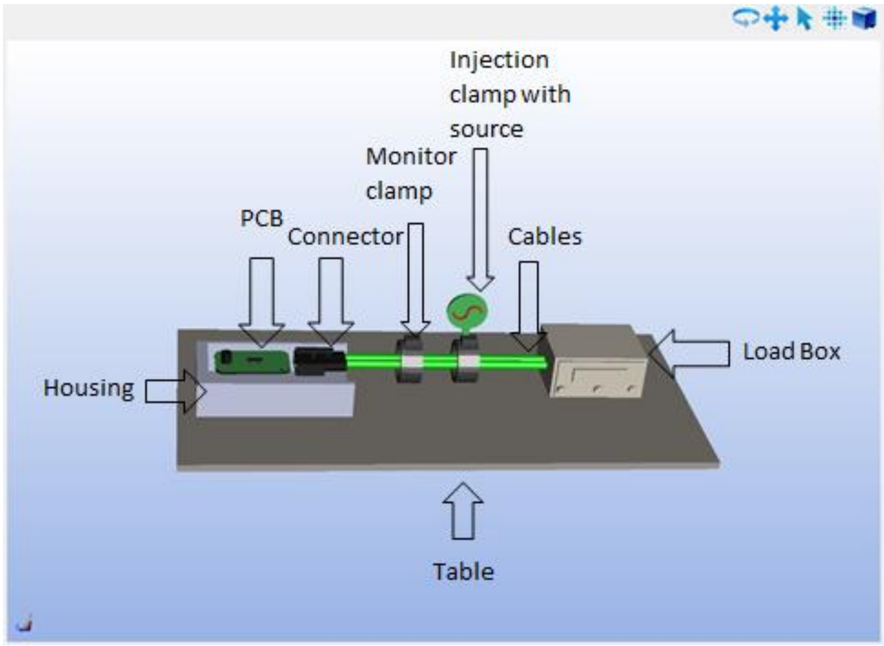


Figure 4-7: BCI experiment apparatuses

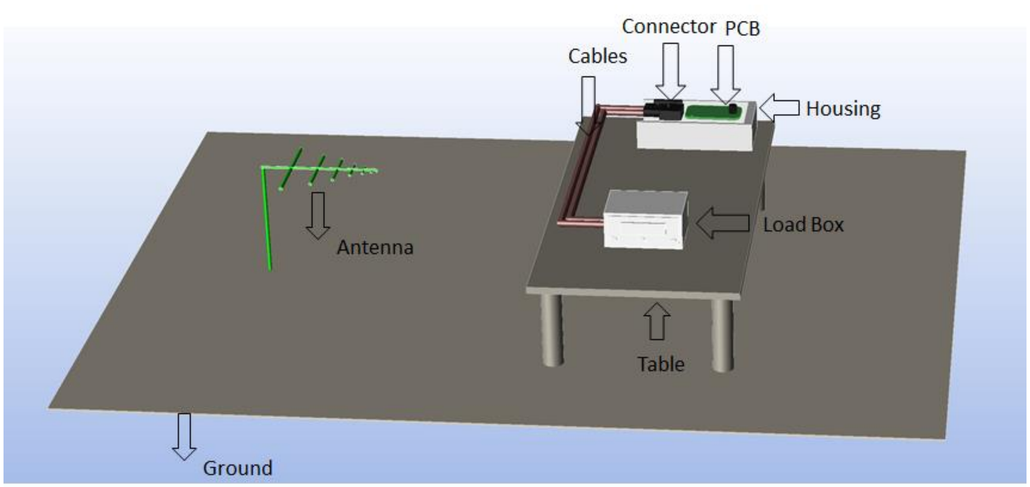


Figure 4-8: RE experiment apparatuses

There are only 2 operations permitted on an Apparatus, as shown in Figure 4-9:

(a) Open Apparatus and

(b) Delete Apparatus

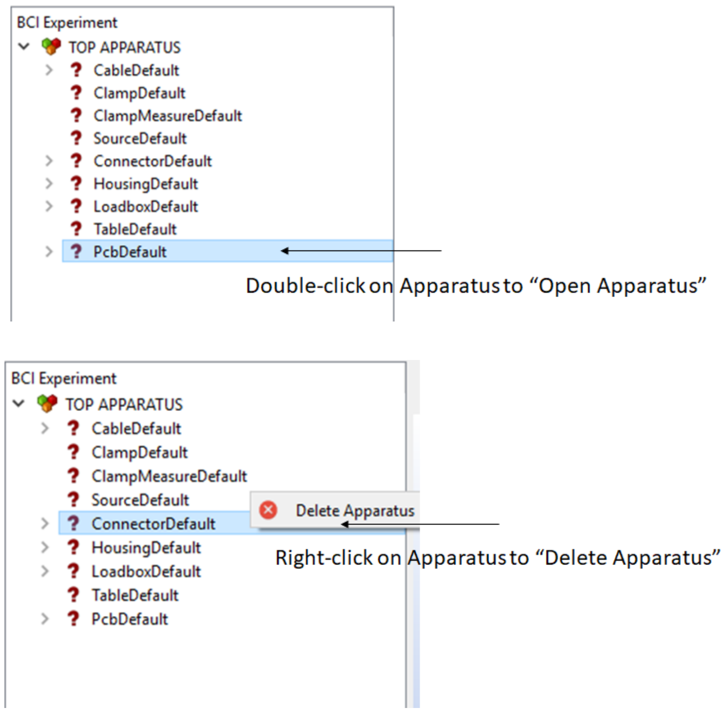


Figure 4-9: Apparatus operations from the apparatus-list sub-screen

Note: “Open Apparatus” cannot be performed for the “Table” “Source” and “Antenna” apparatuses. For the latter two apparatuses there is no model requirements – hence the “Open Apparatus” is useless. “Delete Apparatus” can be performed only for “Connector” and “Housing” apparatuses.

“Open Apparatus” is aimed to import a model, either electrical or physical, for the apparatus. It can be performed from the “Experiment apparatus list” sub-screen by double-clicking on any given apparatus. The same can be achieved by double-clicking on the corresponding apparatus visual model. The methods to import and edit the model for each apparatus is given in Chapter 6: Individual Apparatus.

In Table 3: Apparatus model types, we provide a summary of the model types supported for each apparatus:

Table 3: Apparatus model types

Apparatus Name	Model Type	Model Supported	Extra Information
Cable	Physical	1. GUI drawing	Cable length
Injection Clamp	Electrical	1. Import s3p files 2. Library model	Position on cable
Measurement Clamp	Electrical		Position on cable
Source	Default		
Connector	Physical	1. Import 3D geometry (iges, step, sat)	
	Electrical	1. Import SPICE/Touchstone file (.cir, .sp, .s*p)	
Housing	Physical	1. Import 3D geometry (iges, step, sat)	
	Electrical	1. Import SPICE/Touchstone file (.cir, .sp, .s*p)	
Load-box	Electrical	1. Import SPICE/Touchstone file (.cir, .sp, .s*p)	
	Physical	Will be supported in a future version	
Table	Default		Height from table
Antenna	Default		
PCB	Physical	1. ODB++	

“Delete Apparatus” will remove the apparatus visual model from the 3D visualization screen and will update the apparatus interfaces of the apparatuses that were previously connected to the deleted apparatus. For example, on deleting the connector apparatus, the resultant apparatus list and the 3D visualization is shown in Figure 4-10:

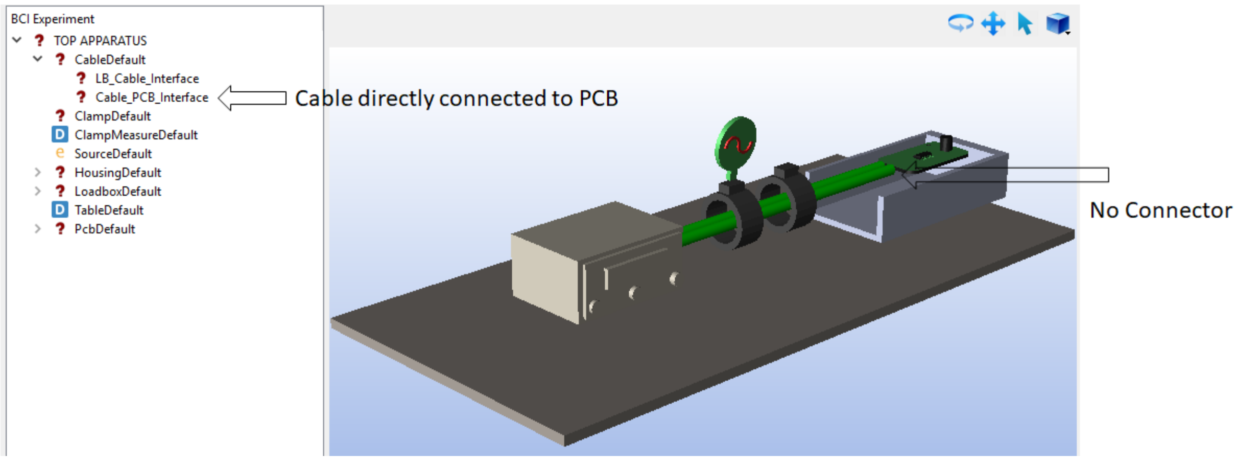


Figure 4-10: Changes on deletion of connector apparatus

Once a model is loaded for each apparatus, there will be an indication of the type of model, in the experiment apparatus list sub-screen as shown in Figure 4-11.

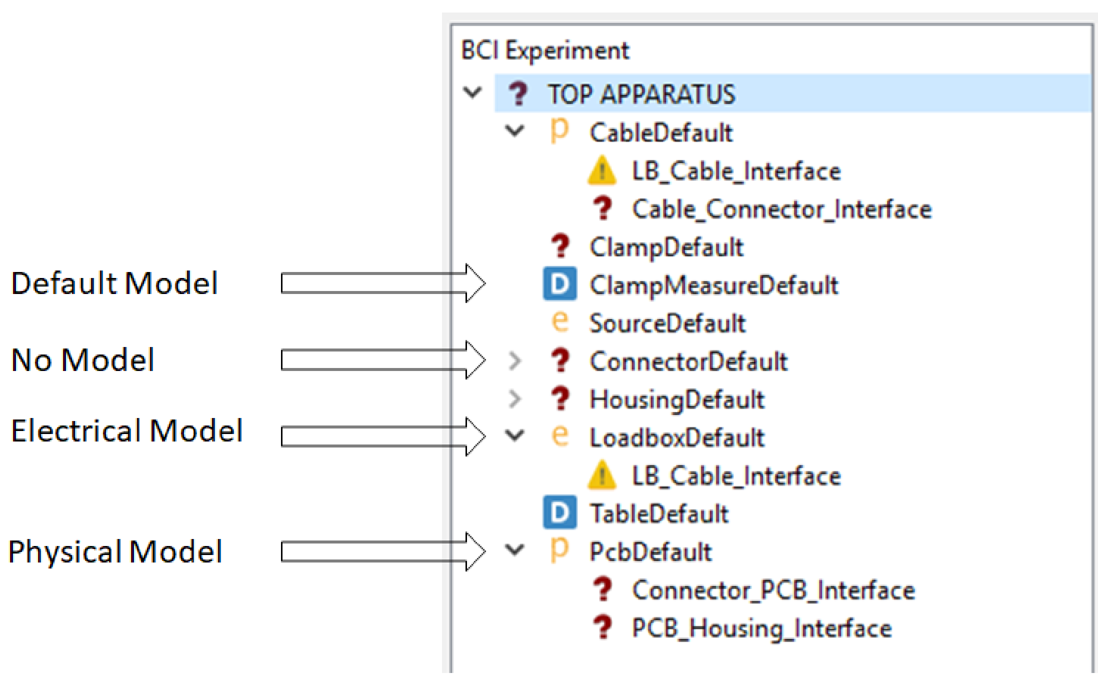


Figure 4-11: Model indications for apparatuses

In summary the icons next to each apparatus can be interpreted as shown in:

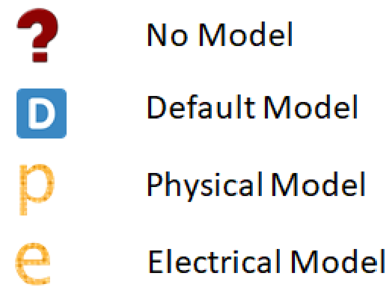


Figure 4-12: Model type interpretation for apparatus models

4.2.2 Apparatus Interfaces

The apparatus interfaces refer to the wiring between apparatuses in an experiment setup. Each apparatus has a list of apparatus-interfaces when expanded in the tree-view **Error! Reference source ot found.**. The icon next to an apparatus-interface refers to the state of the interface. The possible states are shown in Figure 4-13.

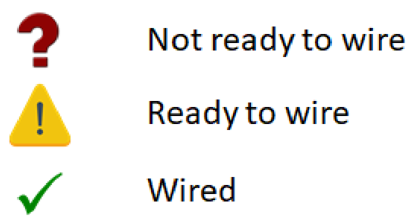


Figure 4-13: Apparatus-interface states

For example, a typical state of apparatus-interfaces during an experiment construction is shown in Figure 4-14.

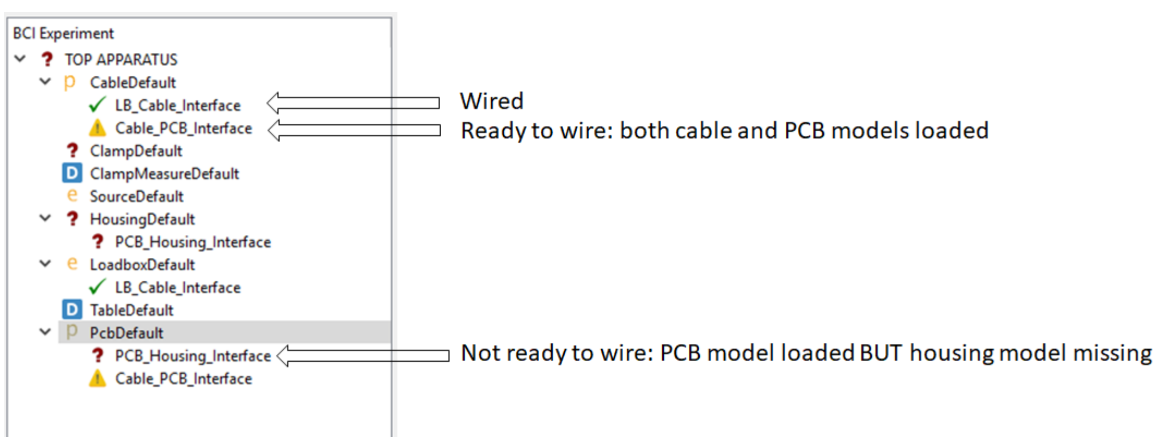


Figure 4-14: Example of apparatus-interfaces in different states

When an apparatus-interface is ready, the wiring can be edited by double clicking the corresponding apparatus-interface, as shown in Figure 4-15.

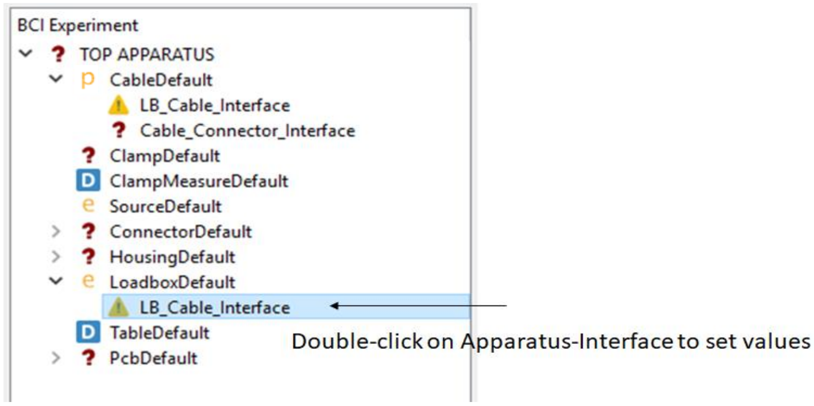


Figure 4-15: Opening an apparatus interface for wiring

More details about apparatus interfaces can be found in Options Tab

The options tab contains features that control the parameters for the experiment. It consists of the following tabs:

(a) BCI options OR RE options

(b) Analysis options

(c) Mesh options

(d) Analysis results

4.2.3 Results

The “Results Toolbox” is only populated after a successful analysis step as shown in Figure 4-16.

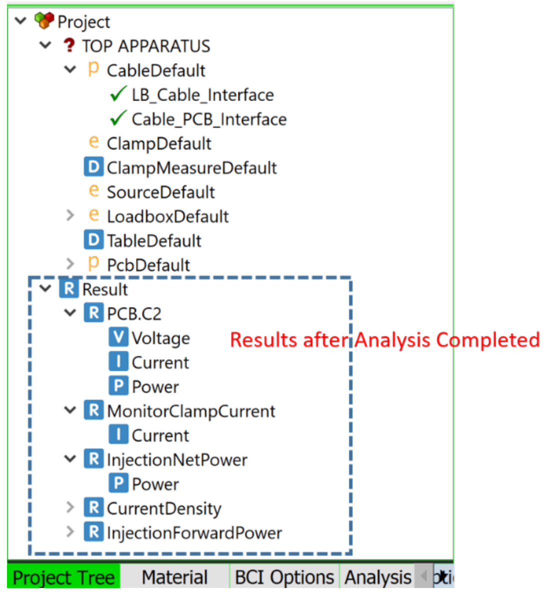


Figure 4-16: Results after analysis

Please refer to **Error! Reference source not found. Error! Reference source not found.** for operational tails of this tab.

4.3 BCI options

This tab contains the different BCI options as shown in Figure 4-17:

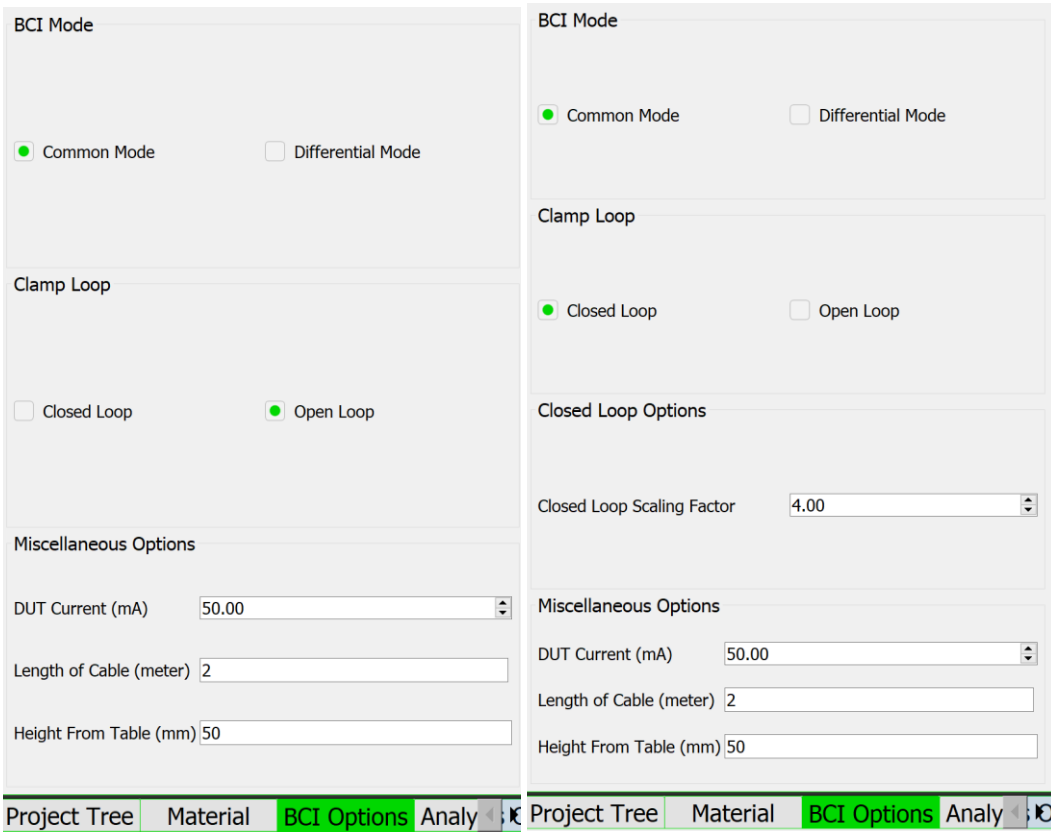


Figure 4-17: BCI options

Currently the latest version of CompScope supports both “Common Mode” BCI and “Differential Mode” BCI. When “Differential mode” is enabled, the user can select which cables are inside or outside the clamp in the “Cable Apparatus Window”.

In terms of clamp conditions, both “Open Loop” (OR “Substitution Method”) and “Closed Loop” are supported.

The “DUT current” is used for internal calibration of the power source fed into the injection clamp.

The “Closed Loop Scaling Factor” is only applicable for “Closed Loop” simulations and is used for computing the input power.

4.4 RE options

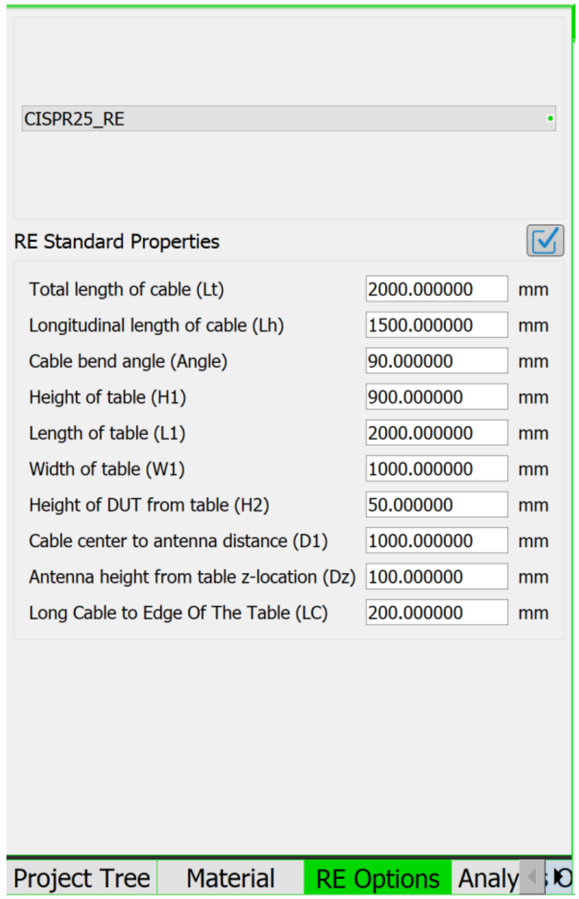


Figure 4-18: RE Options

Currently only CISPR25 standard is supported under RE options. The symbols are explained in Figure 4-19: CISPR 25 options dimension symbols

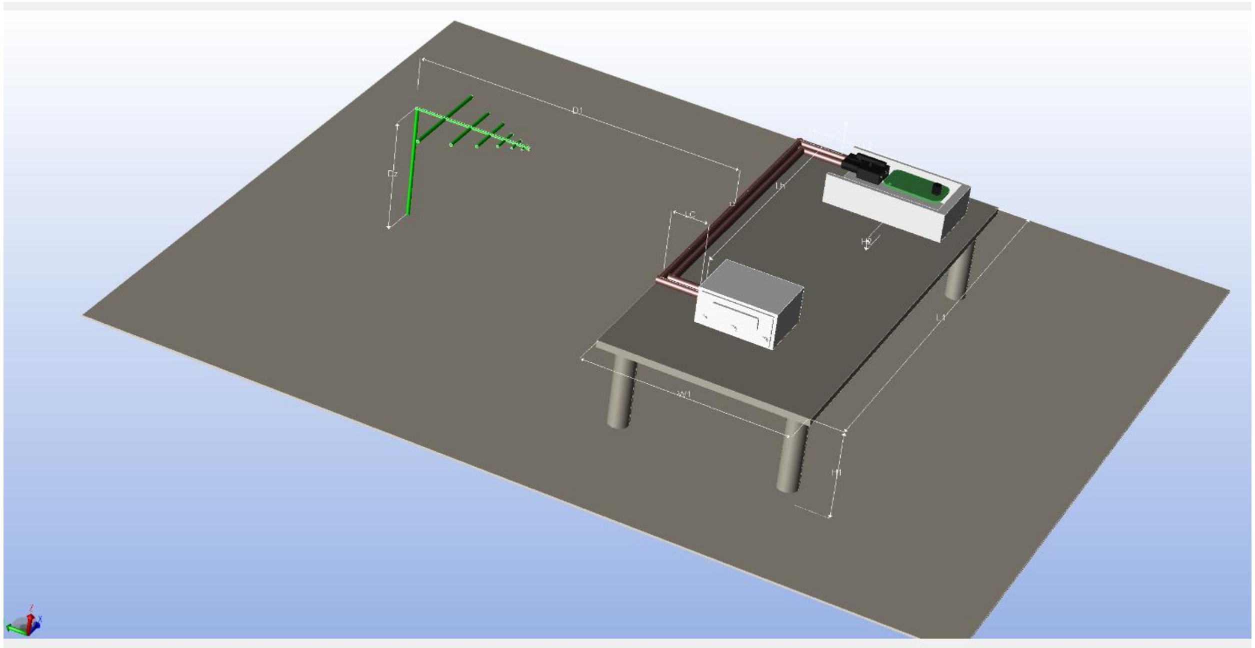


Figure 4-19: CISPR 25 options dimension symbols

4.5 Analysis options

The Analysis Options can be decomposed into 4 distinct parts as shown in Figure 4-20:

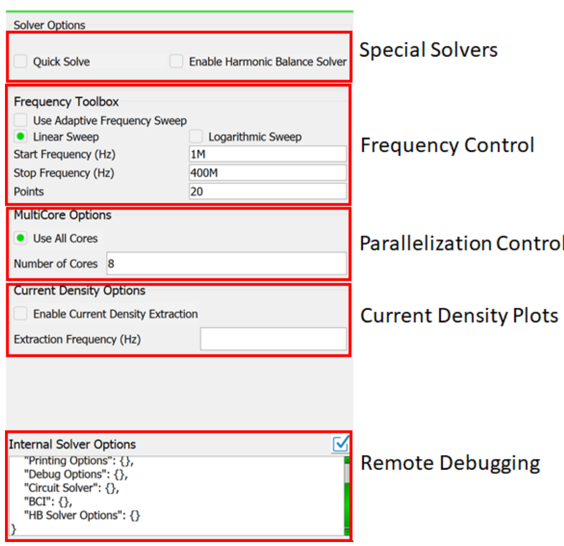


Figure 4-20: Different components of Analysis Options

4.5.1 Special Solver: Quick Solve

Quick Solve is an option that when enabled expedites the solve time by 10-20x but some accuracy is sacrificed. This option should only be used for trend analysis. One good example is when the user needs to understand the positive or negative implications of changing design parameters.

4.5.2 Special Solver: Harmonic Balance

Harmonic Balance is currently specific to BCI experiments. Many BCI failures occur due to malfunction of discrete active components embedded on the PCB. The Harmonic Balance approach can be used in conjunction with electromagnetic solvers to accurately simulate such RF immunity behavior without compromising on the non-linear response of the active elements.

Using Harmonic Balance, CompScope-BCI can generate results like the current through non-linear elements or voltage across non-linear elements under the BCI setup. Therefore failure in discrete non-linear elements on the PCB due to the current injection can be easily flagged as pass/fail. The results for Harmonic Balance can be found in 10.3 BCI results: Harmonic Balance.

Technology overview: The non-linear elements are modelled by means of a truncated Fourier series which is expressed in circuits as sinusoidal voltage/current sources. These voltage sources are then introduced to the linear portion in place of the non-linear elements at the shared ports and utilized to compute the linear current. Similarly, the non-linear current is computed by running a transient simulation on the combination of sinusoidal source and the non-linear elements. A solution is achieved when the linear and non-linear currents are balanced thereby reducing the KCL error at the shared ports. Therefore, the underlying problem is an optimization one, wherein the cost function to be minimized is the KCL error. The most popular algorithm used in Harmonic Balance to minimize the KCL error is Newton-Raphson.

4.5.3 Frequency Control options

This sub-screen enables the user to choose the start-frequency, end-frequency and the number of frequency points either in log or linear scale. The start and end frequencies are common to both log and linear sweep. The difference lies in choosing the frequency step-size. For linear sweep the step-size is decided by the total number of frequency points requested in “Points” text-box. For log sweep the step-size is decided by the entry in “Points per decade”.

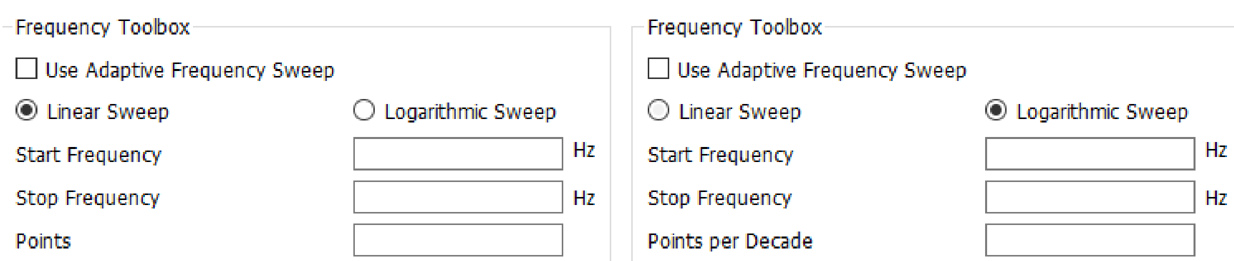


Figure 4-21: Log and linear sweep type inputs

The adaptive frequency sweep option when clicked enables the solver to internally choose, based on a proprietary algorithm, selected frequency points which are explicitly simulated. The output results will still be populated for all frequency points that the user specified using the linear/log sweep parameters.

4.5.4 Parallelization Control options

The parallelization options control the use of cores in the system during “Analyze”. It must be noted that only the 3D full-wave solver is architected to be fully parallel. The 2D solver and the circuit solver run using only 1 thread. This is because the bottleneck in simulation time is the 3D solver. The fraction of time taken by the 2D solver and the circuit solver compared to the entire analysis time is negligible.

By default, the “Use All Cores” checkbox is checked. Under this condition, the software automatically detects the number of processors in the system and populates the “Number of cores”. The “Number of Cores” cannot be edited when the “Use All Cores” has been checked. In order to specify the number of cores the software can use, please uncheck the “Use All Cores” check-box and edit the “Number of Cores” textbox.

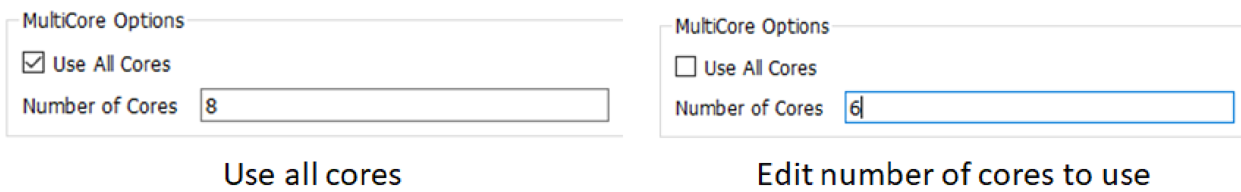


Figure 4-22: Parallelization Options

4.5.5 Enabling current density plot

The “Enable Current Density Extraction” needs to be checked if the user desires to plot the current on the 3D solved structure. The frequency at which this needs to be computed can be entered in the text-box: “Single Frequency for Current Density”. Currently we support computing the current density at a single frequency point only during a frequency sweep analysis. Also, the single frequency should be either the start or the end frequency. These limitations may be removed in future versions based on user requests. In order to show the current density after analysis use the “Results” in the Project Tree.

4.5.6 Remote Debugging options (used by Simyog Application support team)

This field is used by Simyog Application Support team to debug problems at user-site remotely.

4.6 Mesher options

The “Mesher Options” can be decomposed into 3 parts as shown in Figure 4-23.

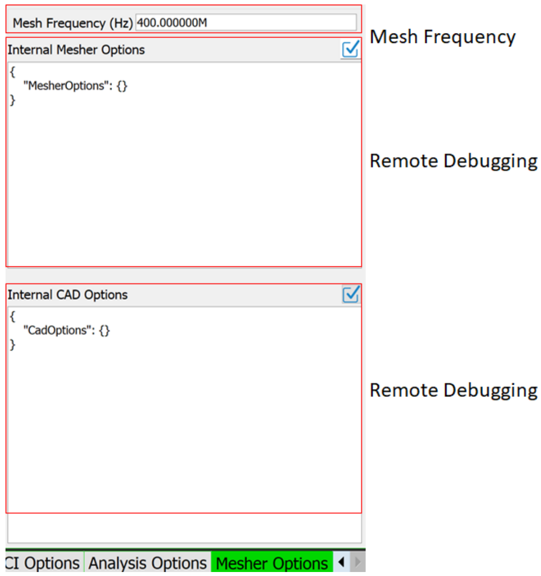


Figure 4-23: Components of Mesher Options

4.6.1 Mesh Frequency

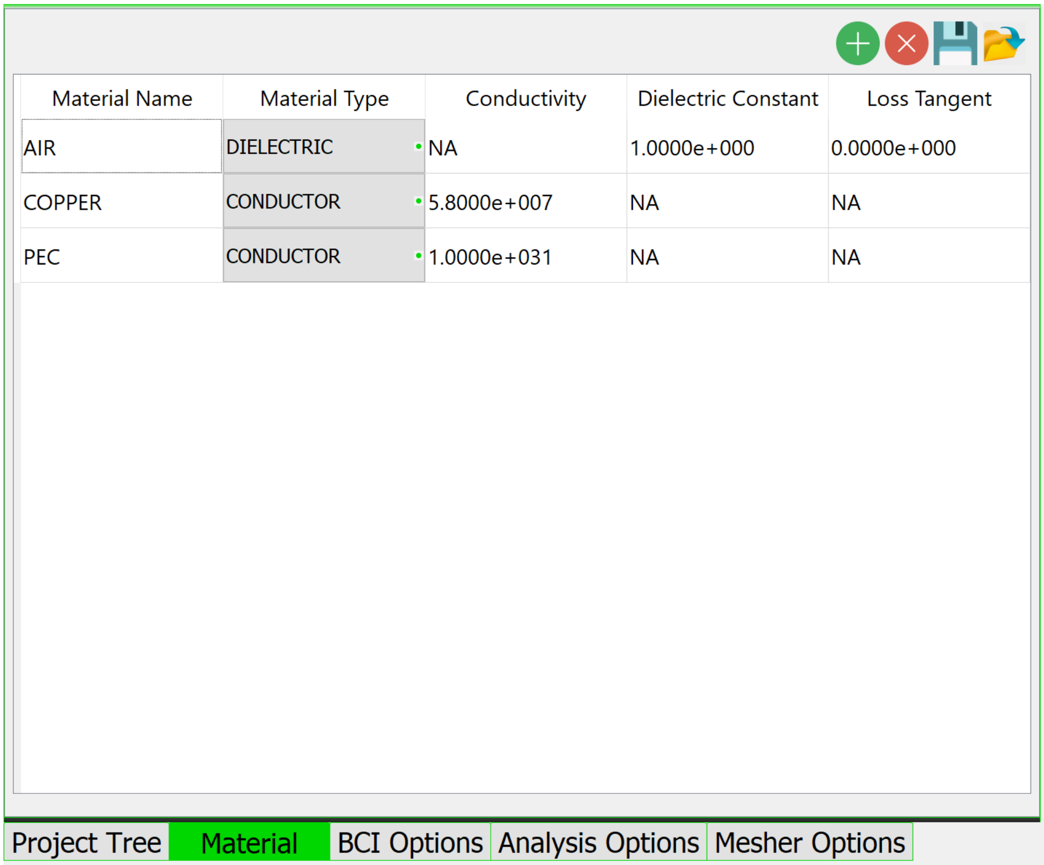
This input is used to determine the maximum element size of a mesh element (triangle). It is advisable to use the maximum or end-frequency of the solver options as the mesh frequency. The maximum element size is determined as the corresponding wave-length divided by 10.

4.6.2 Remote Debugging Support

This sub-screen is used by Simyog Technical Support for remote debugging.

5 Chapter 5: Materials

In this version of CompScope, only Conductors and Non-Dispersive Dielectric (NDD) materials are supported. Material list table, shown in the following figure, contains all the materials used in the experiment. The Material list table is a part of the BCI Main Window Properties Box. All materials listed here can be accessed by the Individual Apparatus Windows.



Material Name	Material Type	Conductivity	Dielectric Constant	Loss Tangent
AIR	DIELECTRIC	NA	1.0000e+000	0.0000e+000
COPPER	CONDUCTOR	5.8000e+007	NA	NA
PEC	CONDUCTOR	1.0000e+031	NA	NA

Figure 4-1: Material List Table

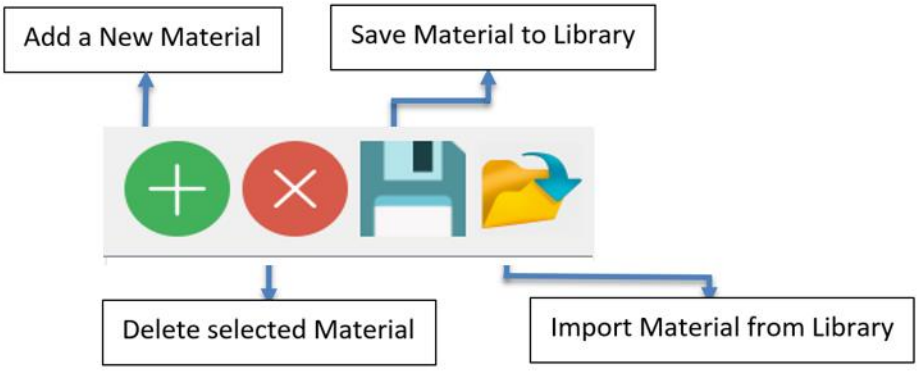


Figure 4-2: Material List Toolbar

5.1 Material Properties

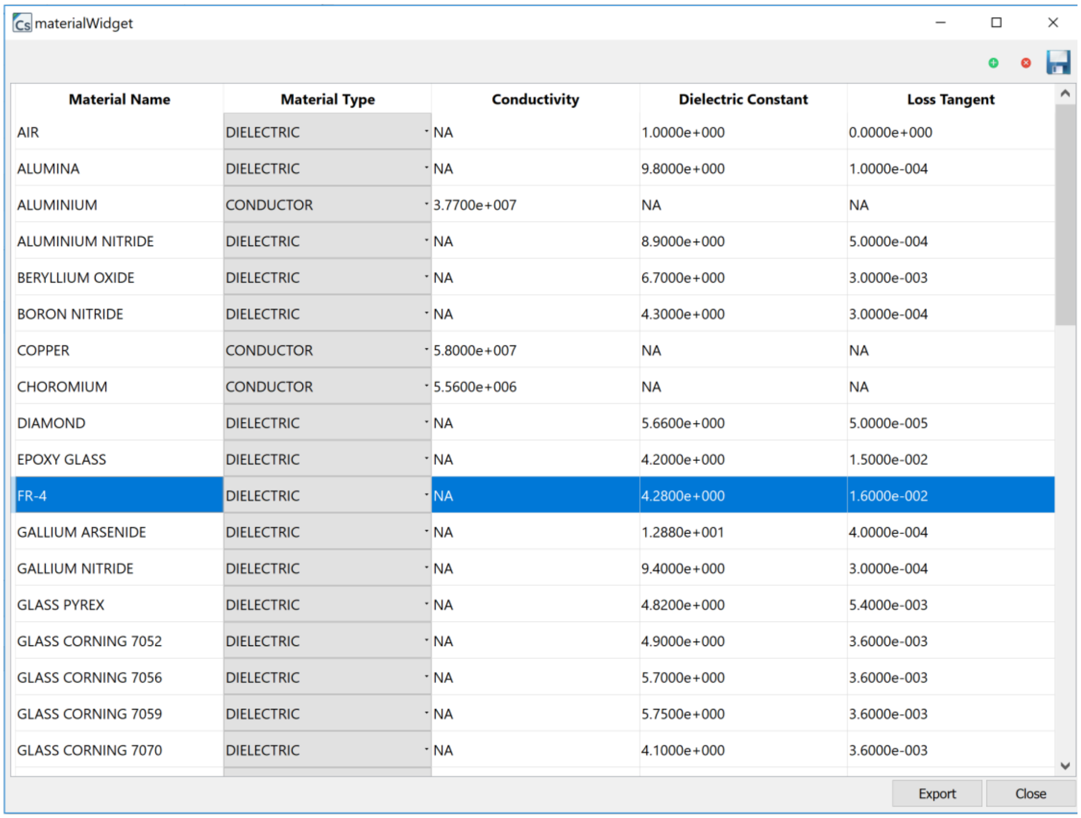
Conductor materials are characterized by “Conductivity”. Other entries will be “NA”.

Dielectric materials are characterized by Dielectric Constant and Loss Tangent OR Dielectric Constant and conductivity. If user enters Loss tangent then the conductivity entry will automatically become “NA” and if user enters conductivity the Loss Tangent entry will automatically become “NA”.

New materials can be added and saved to the library for future use. Material names are currently not editable, but this feature will be supported soon.

5.2 Material Library

Compscope Material Library has all the standard materials. One can import any material from the library by clicking , selecting the required material and press export button. Also a new material can be added by clicking button.



Material Name	Material Type	Conductivity	Dielectric Constant	Loss Tangent
AIR	DIELECTRIC	NA	1.0000e+000	0.0000e+000
ALUMINA	DIELECTRIC	NA	9.8000e+000	1.0000e-004
ALUMINIUM	CONDUCTOR	3.7700e+007	NA	NA
ALUMINIUM NITRIDE	DIELECTRIC	NA	8.9000e+000	5.0000e-004
BERYLLIUM OXIDE	DIELECTRIC	NA	6.7000e+000	3.0000e-003
BORON NITRIDE	DIELECTRIC	NA	4.3000e+000	3.0000e-004
COPPER	CONDUCTOR	5.8000e+007	NA	NA
CHOROMIUM	CONDUCTOR	5.5600e+006	NA	NA
DIAMOND	DIELECTRIC	NA	5.6600e+000	5.0000e-005
EPOXY GLASS	DIELECTRIC	NA	4.2000e+000	1.5000e-002
FR-4	DIELECTRIC	NA	4.2800e+000	1.6000e-002
GALLIUM ARSENIDE	DIELECTRIC	NA	1.2880e+001	4.0000e-004
GALLIUM NITRIDE	DIELECTRIC	NA	9.4000e+000	3.0000e-004
GLASS PYREX	DIELECTRIC	NA	4.8200e+000	5.4000e-003
GLASS CORNING 7052	DIELECTRIC	NA	4.9000e+000	3.6000e-003
GLASS CORNING 7056	DIELECTRIC	NA	5.7000e+000	3.6000e-003
GLASS CORNING 7059	DIELECTRIC	NA	5.7500e+000	3.6000e-003
GLASS CORNING 7070	DIELECTRIC	NA	4.1000e+000	3.6000e-003

Figure 4-3: Material Library

6 Chapter 6: Individual Apparatus Window

6.1 Cable Apparatus

The Cable Apparatus/widget can be opened by either of the two ways:

Go to “CableDefault” option from the Apparatus List described in 0

a) Project Tree. Double click on CableDefault -> Open Apparatus. The Cable Widget opens up.

b) Click “Select” button from Visualization window of BCI main window and then click on the Cables in the Experiment Main screen.

The following video explains the operations on the cable widget:

http://simyog.com/resources/manual_animations/CableDefault.mp4

The figure below shows a Cable Widget Window:

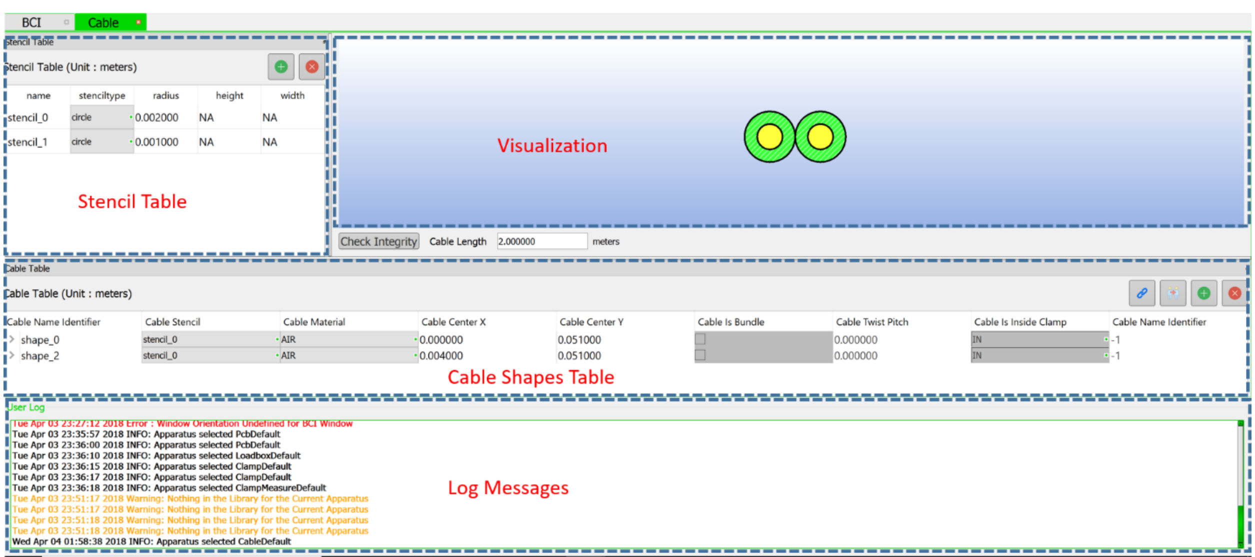


Figure 6-1: Cable Widget Window

6.1.1 Utility Options

Load, Save and Library options for “Cable” apparatus are not supported in this version of CompScope.

6.1.2 Stencil Table

Stencil table contains all the stencils used for cable harness table. A stencil is a shape without a center. Presently only circle stencil is supported.

Stencil Table (Unit : meters)

name	stenciltype	radius	height	width
stencil_0	circle	0.002000	NA	NA
stencil_1	circle	0.001000	NA	NA

Figure 5-2: Stencil Table

6.1.3 Cable Shapes Table

Presently Compscope supports two types of cables

- Straight Cables.
- Bundle of twisted cables with user defined twist pitch.

Support for shielded cables will be available in a later release.

6.1.3.1 Cable Harness Toolbar Description:

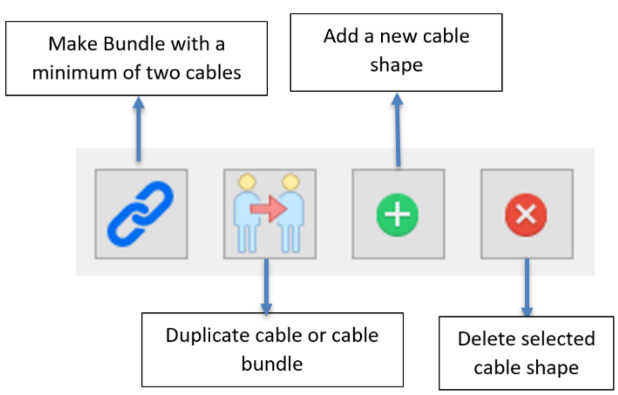


Figure 5-3: Cable Harness Toolbar

6.1.3.2 Straight Cables:

Step 1: Add a shape by clicking button. Select the stencil type, material and edit the Cable center X and Cable center Y. A new shape shape_0 is created. The following figure shows the cable widget window after Step 1.

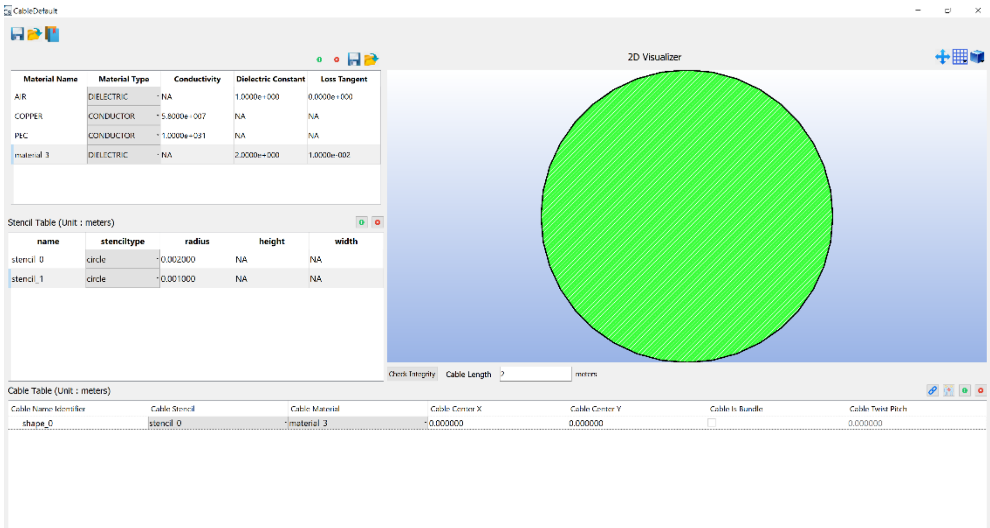


Figure 5-4: Cable Widget Window after Step 1

Step 2: Select shape_0 and click button to add a shape inside the selected shape (here it is shape_0). On clicking shape_0, shape_1 will be visible. Select the stencil type, material and edit the Cable Center X and Cable Center Y for shape _1. Note that the outermost shape cannot be a Conductor material. The following figure shows the cable widget window after Step 2.

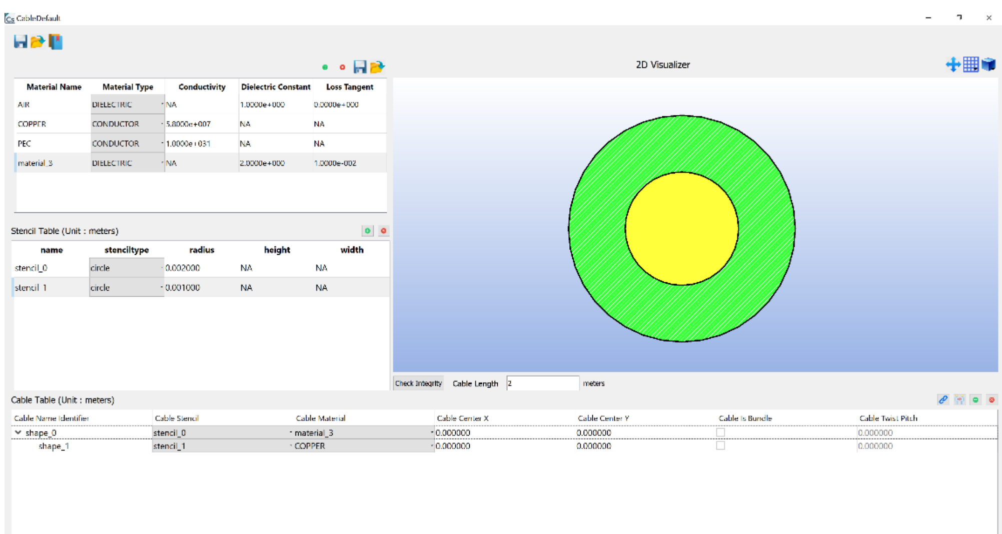


Figure 5-5: Cable Widget Window after Step 2

Step 3: For adding one more cable either follow steps 1 and 2 or click on replicate/clone button . Replication is explained in 6.1.3.4 Replication.

6.1.3.3 Twisted cables:

Twisted cables are only supported in BCI experiment currently. It will soon be available for RE. Twisted cable bundle can be enabled by first adding straight cables (at least 2) and making a bundle and then specifying the twist pitch. The following figure shows the cable widget window after 2 straight cables are added.

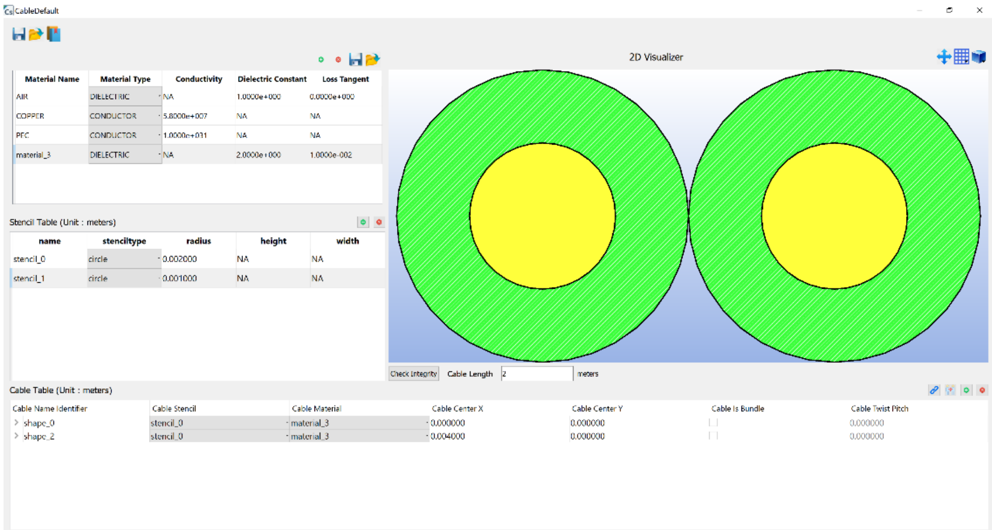


Figure 5-6: Cable Widget Window after adding two straight cables

Now select shape_0 and shape_2 and click on  button to make a bundle. The following figure shows the cable widget window after making a bundle.

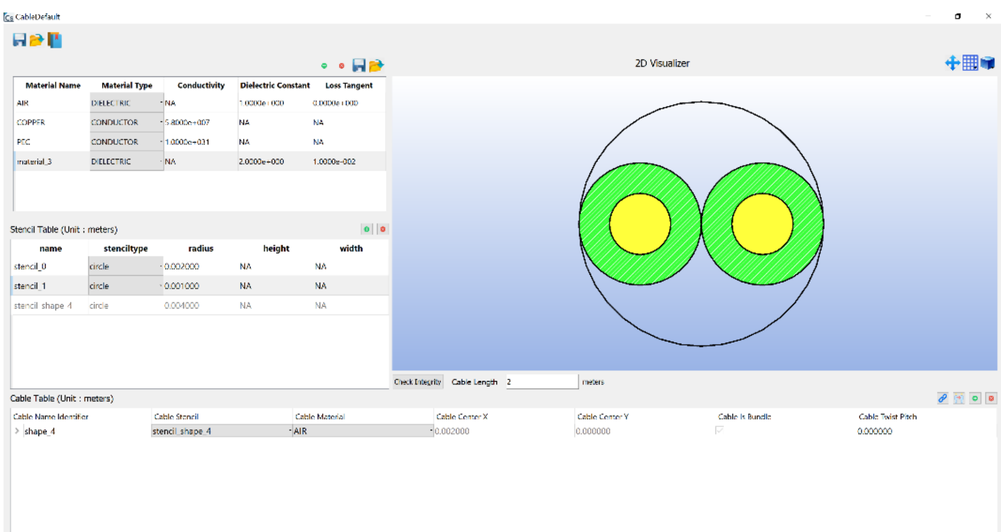


Figure 5-7: Cable Widget Window after making a bundle

After making a bundle, user defined Cable Twist Pitch can be given by editing the Cable Twist Pitch. Note that Cable Twist Pitch is in meters, which is the length of one pitch. The following figure shows the cable widget window after editing the Cable Twist Pitch.

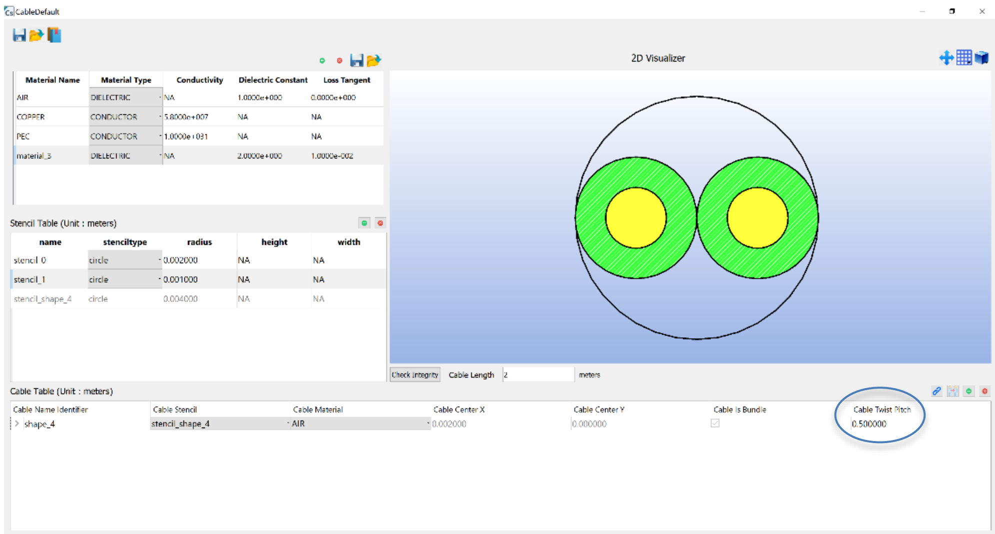


Figure 5-8: Cable Widget Window after entering Twist Pitch

6.1.3.4 Replication

Select the shape to be replicated and click on , enter the number of cable, X-increment and Y increment appropriately and click ok button. The following figures shows the cable widget window after entering the number of cables as 2, X increment as 4e-3 and Y-increment as 0.

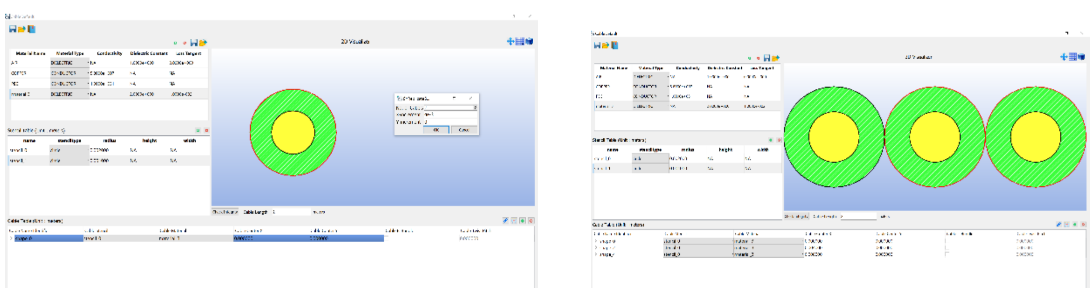


Figure 5-9: Cable Widget Window after replication of cable

6.1.3.5 Deleting shapes and bundles

Shapes and cable bundles can be deleted by selecting the appropriate shape to be deleted and clicking on button. Only one shape or one bundle can be deleted at a time.

6.1.4 2D Visualizer

2D Visualizer is where the cable harness can be seen. It is dynamically populated as you enter a new cable in the cable harness table.

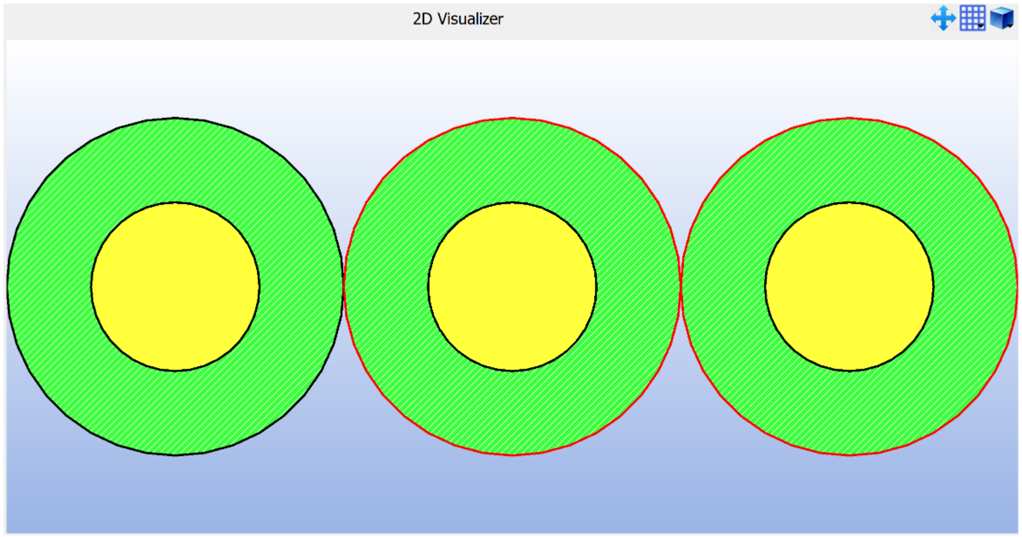


Figure 5-10: 2D Visualizer window showing 3 cables

Grid View shows the grid and also has several options to change units and see the cross section of cable harness in different grid units.

6.1.5 Check Integrity and Cable Length

User defined length of the cable can be given by entering the length of the cable in meters.

Check Integrity button is used to check if there are any intersections in the cable harness. User can check the log message window on the main screen after clicking the check integrity button for any warnings. The following figure shows one such example.

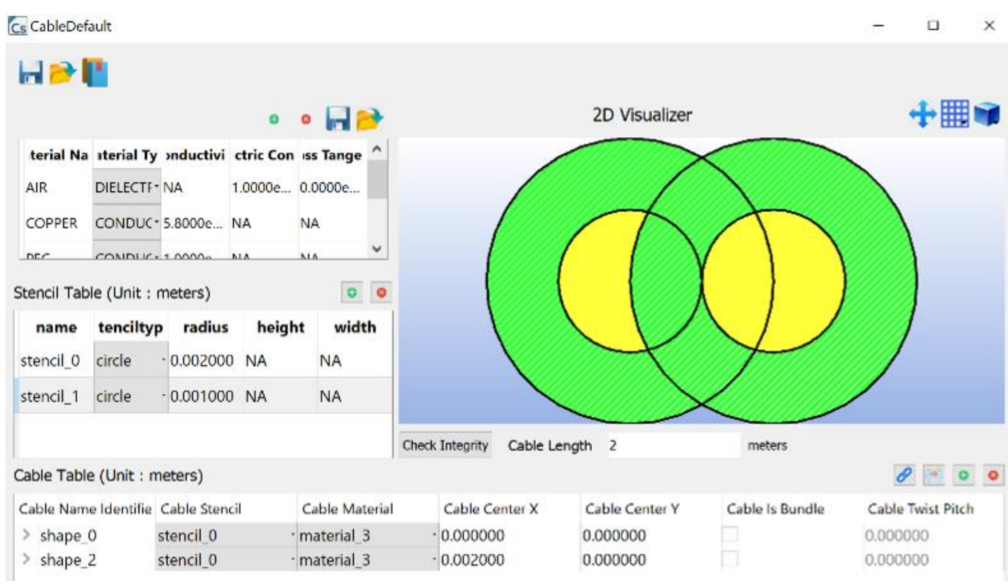
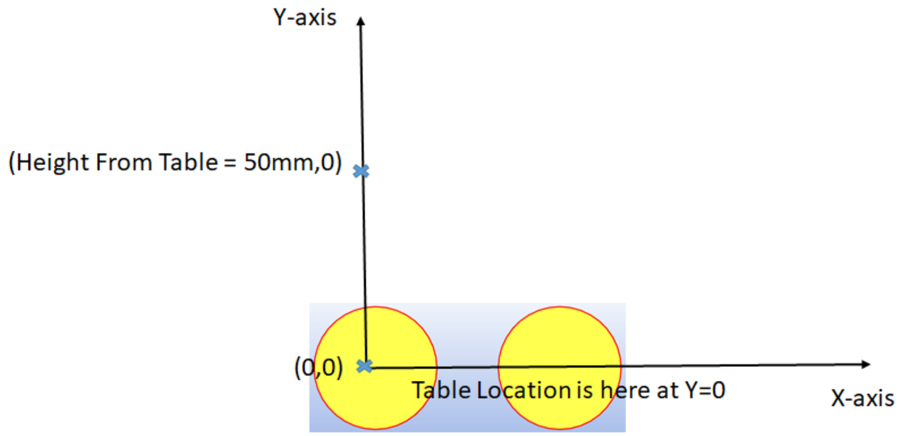


Figure 5-12: Check Integrity Test

The "check integrity" button will automatically shift the cables to the correct location so that the user does not need to manually adjust heights.

BEFORE PRESSING "CHECK INTEGRITY"



Cable Table

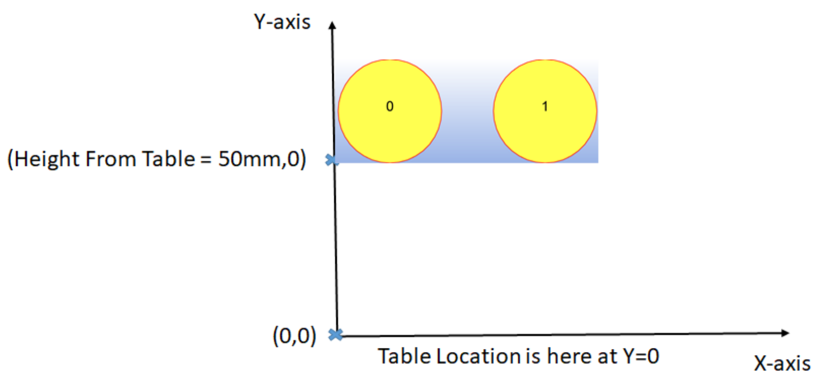
Cable Table (Unit : meters)

Unit is meter

Cable Name Identifier	Cable Stencil	Cable Material	Cable Center X	Cable Center Y
shape_0	stencil_0	COPPER	0.000000	0.000000
shape_1	stencil_0	COPPER	0.006000	0.000000

Figure 6-2: BCI experiment - before clicking the "Check Integrity" button

AFTER PRESSING "CHECK INTEGRITY"



Cable Table (Unit : meters)

Unit is meter

Cable Name Identifier	Cable Stencil	Cable Material	Cable Center X	Cable Center Y
shape_0	stencil_0	COPPER	0.002000	0.052000
shape_1	stencil_0	COPPER	0.008000	0.052000

Figure 6-3: BCI experiment - after clicking the "Check Integrity" button

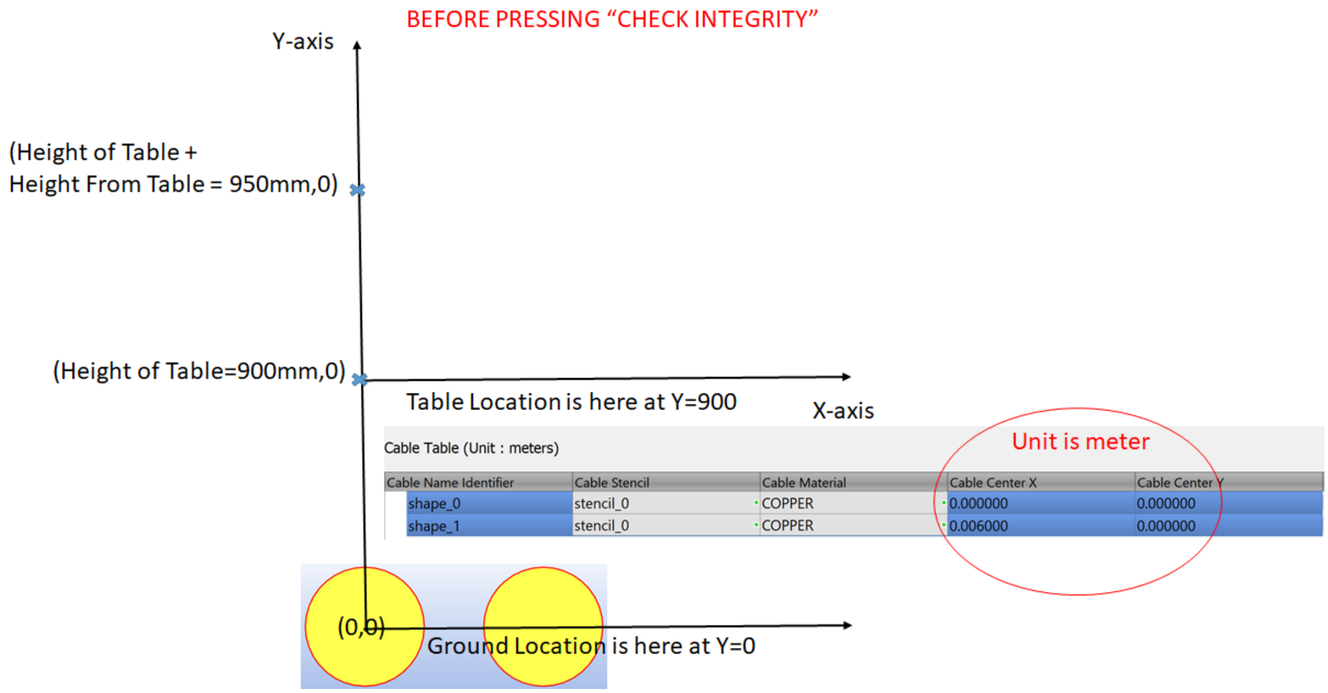


Figure 6-4: RE experiment - before clicking the "Check Integrity" button

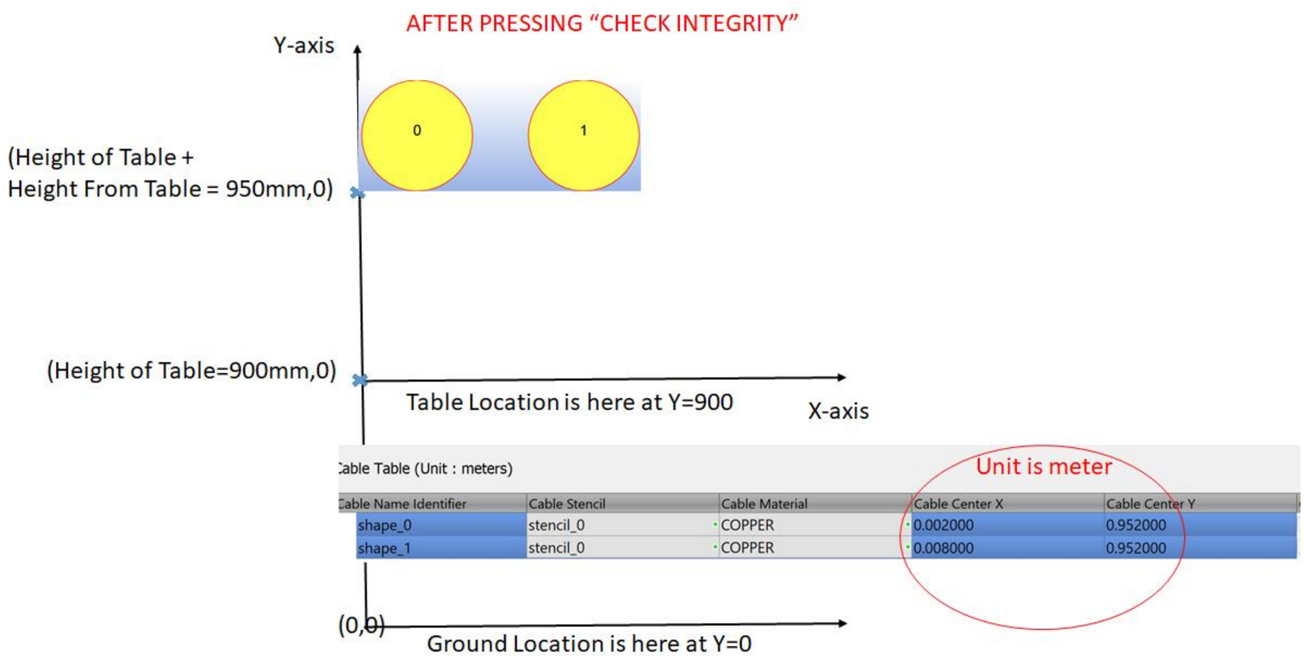


Figure 6-5: RE experiment - after clicking the "Check Integrity" button

6.1.6 Cable numbering for App Interface

If there are ‘N’ cables in the cable harness table they are numbered as n0, n1, n2, …., nN-1. This numbering is used to connect cables to different apparatus. On clicking “Check Integrity” the net number of the cable shapes is computed and demonstrated as shown below:

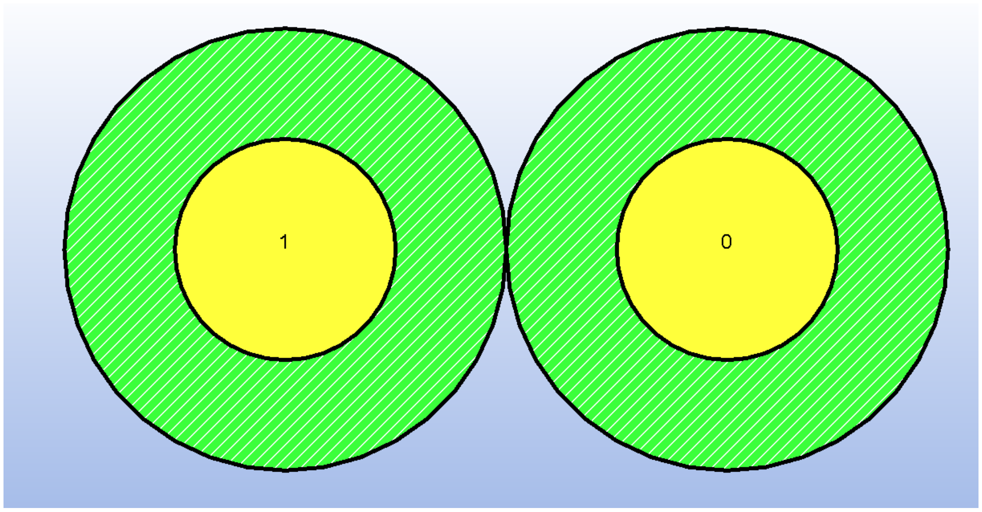


Figure 5-12: Cable Net Numbering

Cable Widget Video: http://simyog.com/resources/manual_animations/CableDefault.mp4

6.2 Clamp Apparatus (BCI only)

The clamp apparatus represents the injection clamp that is used for injecting RF noise to the cable. The clamp apparatus window can be opened by right-clicking the “ClampDefault” from the “Experiment Apparatus List” or the “3D visualization” window in the Experiment screen. The different sections of the clamp apparatus screen is shown in Figure 6-6:

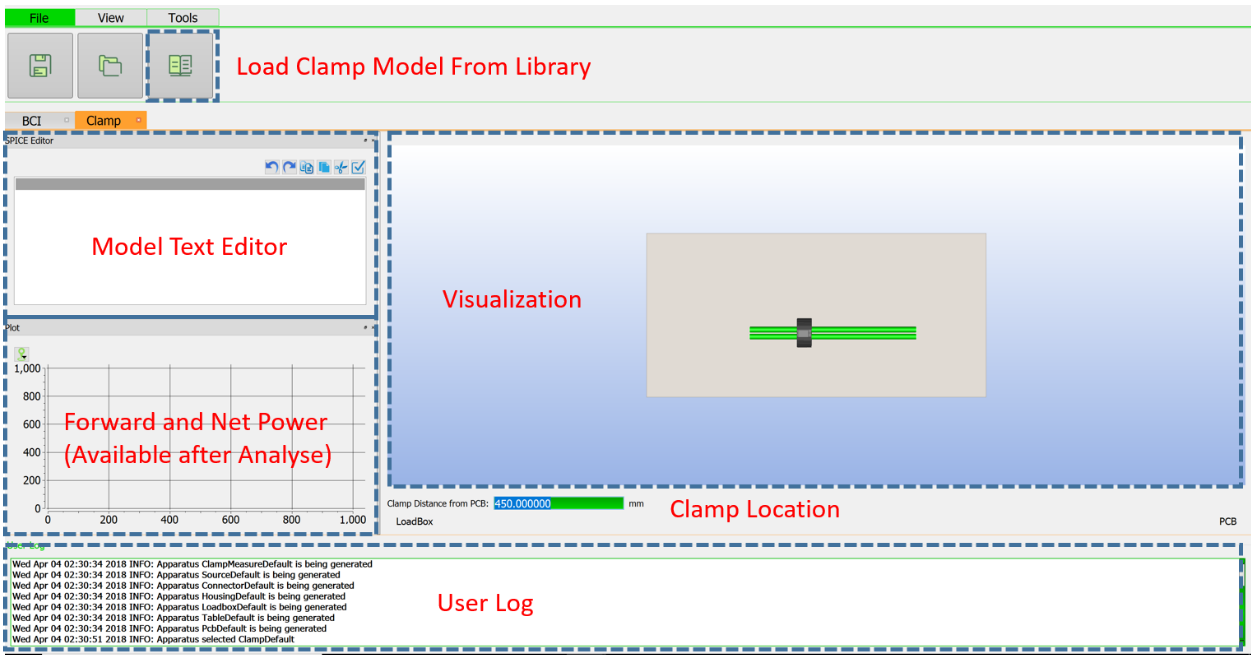


Figure 6-6: Clamp apparatus screen

6.2.1 Load and Save

Load Library Clamp Model: This is the recommended way of loading a clamp model for the experiment. Currently, only F-140 clamp type import is supported. Once loaded, a confirmation will be available on the “Loaded Model Text Box”.

Figure 6-7: Loaded Model Text Box on importing F-140 library model

For more details of the library model, please see 13.3.1 Explanation of CompScope injection clamp library model.

Load Custom Clamp Model: This is a method in which the user can load a custom model, which is not available from the library. The user is prompted to load:

(a) First - an s3p file corresponding to the “clamp-cable coupling model”. Please refer to 13.3.1.1 Injection clamp to cable coupling model

(b) Second – an s3p file corresponding to the “clamp calibration model”. Please refer to 13.3.1.2 Injection clamp calibration model. If no special calibration setting is used, the same model as (a) can be used.

Save Clamp Model: This is not a functional button for this screen

6.2.2 3D Visualization Box

This is 3D visualization box for the clamp with reference to the cable. It gives a visual feedback to the user in terms of placement of the clamp.

6.2.3 Clamp Position Box

This sub-screen enables the user to select the location of the clamp on the cable. The text box can be populated by any number (double) representing the distance in mm from the PCB.

6.2.4 Loaded Model Text Box

This box demonstrates the loaded model for the clamp. For library based model it only mentions the library model name to protect proprietary model information.

6.2.5 Forward and Net Power Box

This sub-screen is only populated after the “Analyze” step is complete. It demonstrates the frequency vs. forward/net power profiles as shown in Figure 6-8.

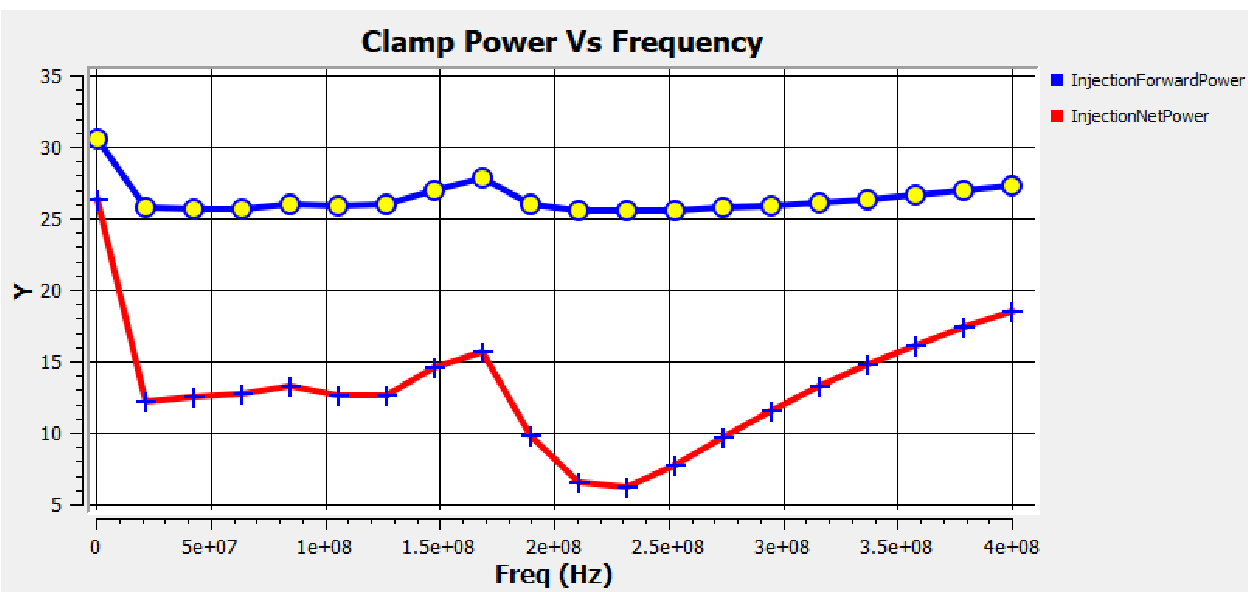


Figure 6-8: Power profiles corresponding to the clamp

The same profile figures can be seen under the “Analysis Results” (refer: **Error! Reference source not ound. Error! Reference source not found.**) as shown in Figure 6-9:

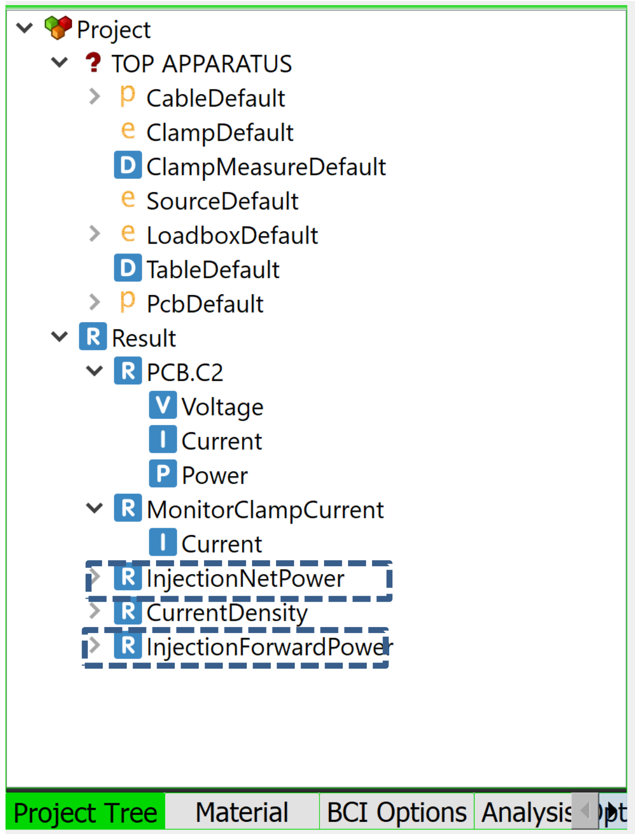


Figure 6-9: Clamp power profiles from the Analysis Result tab

6.3 Measurement Clamp Apparatus (BCI only)

The “Measurement Clamp Apparatus” window is similar to that of the “Clamp Apparatus”, but the functionalities are somewhat different.

6.3.1 Load and Save

There is no model requirement for the measurement clamp. Hence these buttons are purposeless.

6.3.2 3D Visualization Box

This is 3D visualization box for the measurement clamp with reference to the cable. It gives a visual feedback to the user in terms of placement of the measurement clamp.

6.3.3 Measurement Clamp Position Box

This sub-screen enables the user to select the location of the measurement clamp on the cable. The text-box works similar to 6.2.3 Clamp Position Box.

6.3.4 Loaded Model Text Box

Since there is no model requirement for the measurement clamp – this box is also ineffective.

6.3.5 Measurement Clamp Current Box

This sub-screen is only populated after the “Analyze” step is complete. It demonstrates the frequency vs. measured clamp current as shown in Figure 6-10.

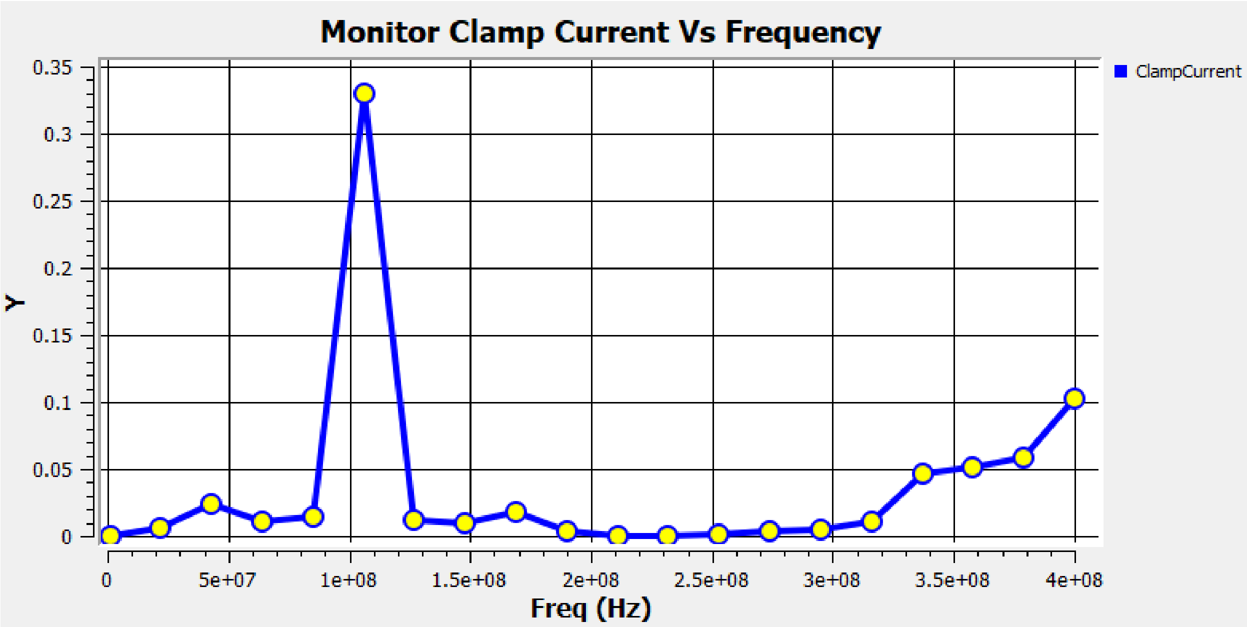


Figure 6-10: Measurement clamp current box

The same profile figures can be seen under the “Analysis Results” (refer: 10.2 BCI results: Component wise Current, Voltage and Power) as shown in Figure 6-11:

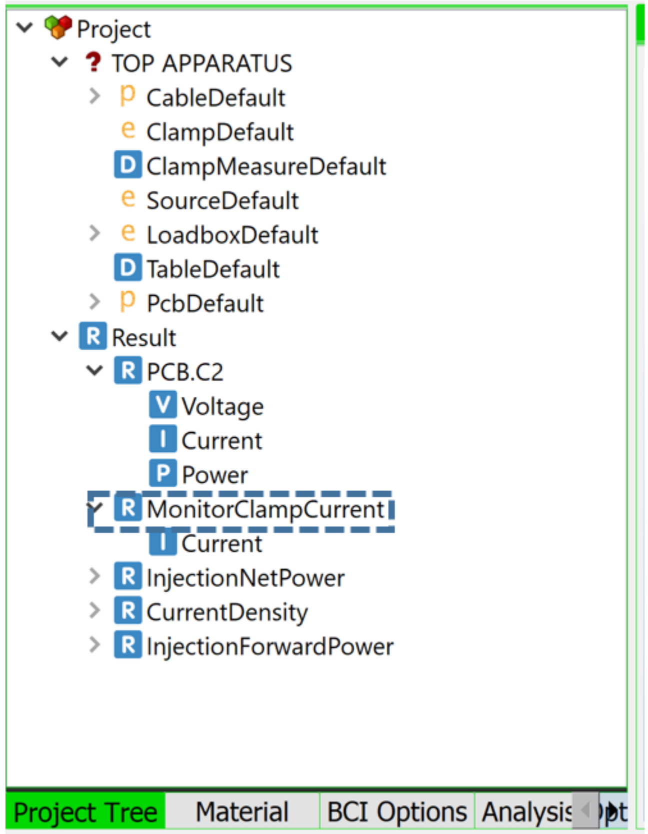


Figure 6-11: Measurement clamp current from Analysis Results

6.4 Source Apparatus (BCI only)

This apparatus window does not contain any information. The source is calibrated using BCI options.

6.5 Loadbox Apparatus

The loadbox comprises of the following components:

(a) VSource: Voltage feed to the PCB through the cables

(b) LISN: Stabilization network for the VSource

(c) Termination: Spice or touchstone file termination for the non-Vsource cables

(d) Ground: Ability to connect schematic spice/touchstone nodes to the table ground

(e) CablePort: Representation of the cable structure for schematic connection

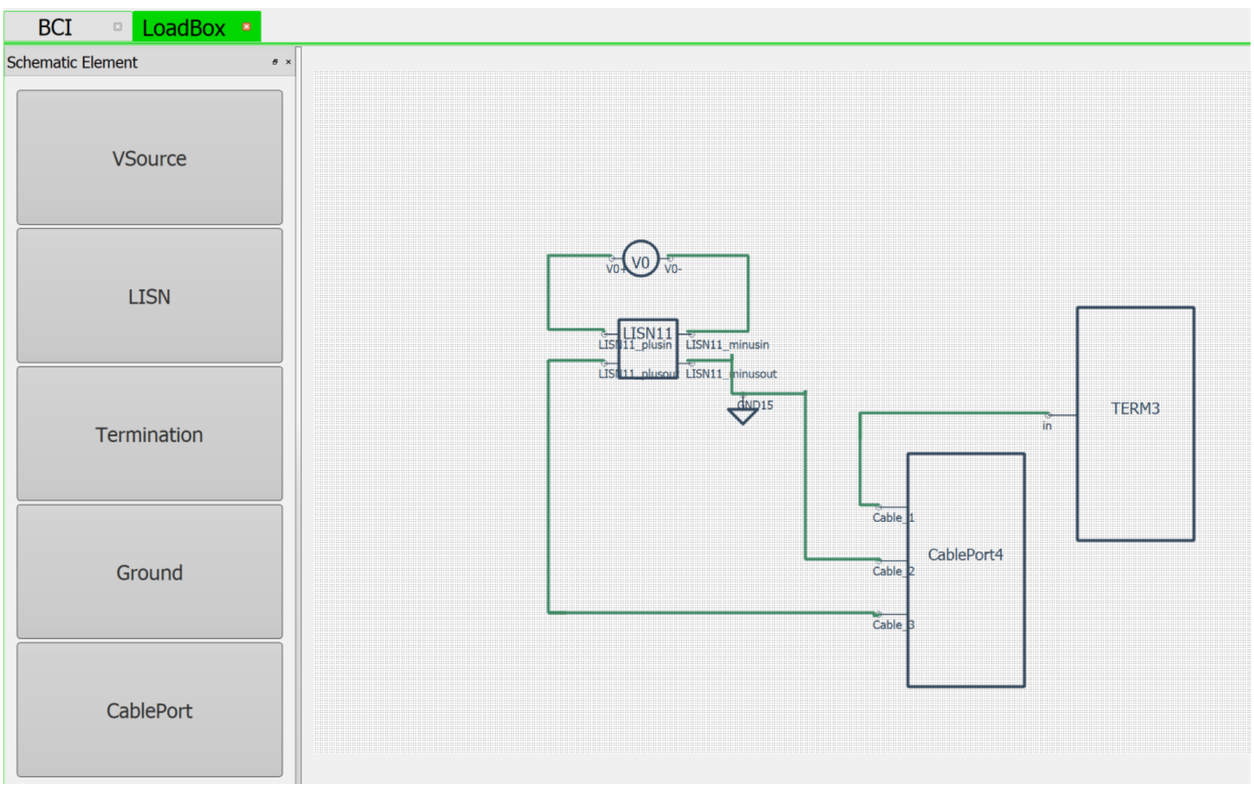


Figure 6-12: Example loadbox setting

Steps to form the loadbox:

(a) Step 1: Drag a CablePort element, right-click and specify the number of cables

(b) Step 2: If required, drag a VSource, right click, click on "Properties" and edit the voltage value as shown below.

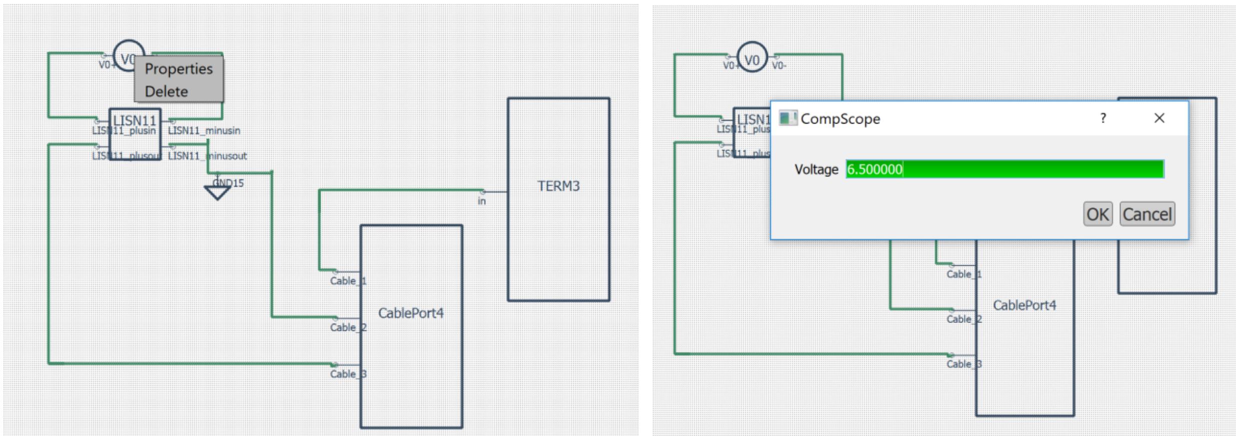


Figure 6-13: Editing the VSource

(c) If required, drag a LISN box, right click, click on "Import SPICE/Touchstone" and then browse the file.

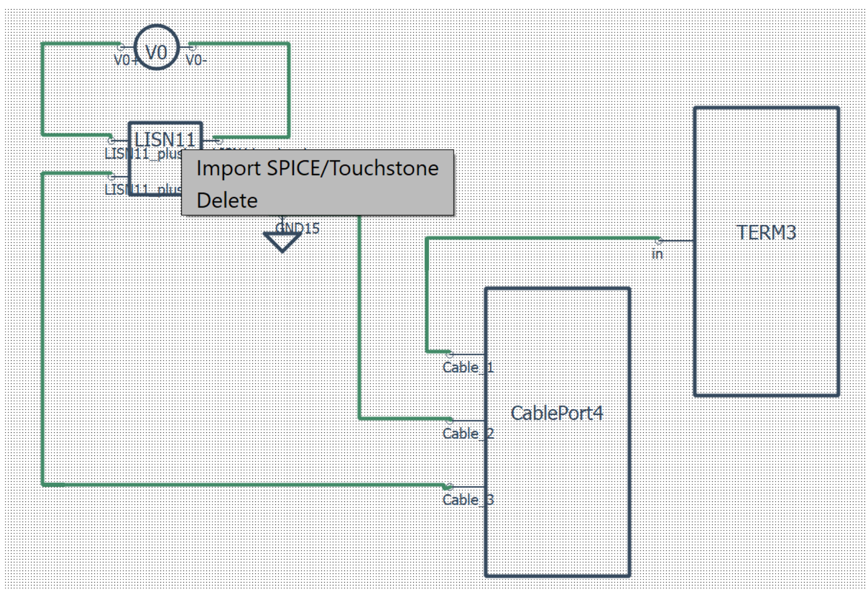


Figure 6-14: Editing the LISN

Note: If a SPICE file is used, please make sure that there is no "0" or "gnd" or "ground" schematic node in the file. Ground or connect to table must be provided using the "Ground" element. Example of a typical LISN file is given below.

```
* demo file
.subckt lisn plusin plusout minusin minusout
C1 plusin minusin1 1e-6
Rc1 minusin1 minusin 1e-3
L1 plusin plusout1 5e-6
Rl1 plusout1 plusout 1e-3
C2 plusout n1 1e-7
R1 n1 minusout 50
R2 minusin minusout 1e-3
.ends
```

Figure 6-15: Example LISN file

(d) If required, drag a "Termination" box, right click, click on "Import SPICE/Touchstone" and then browse the file.

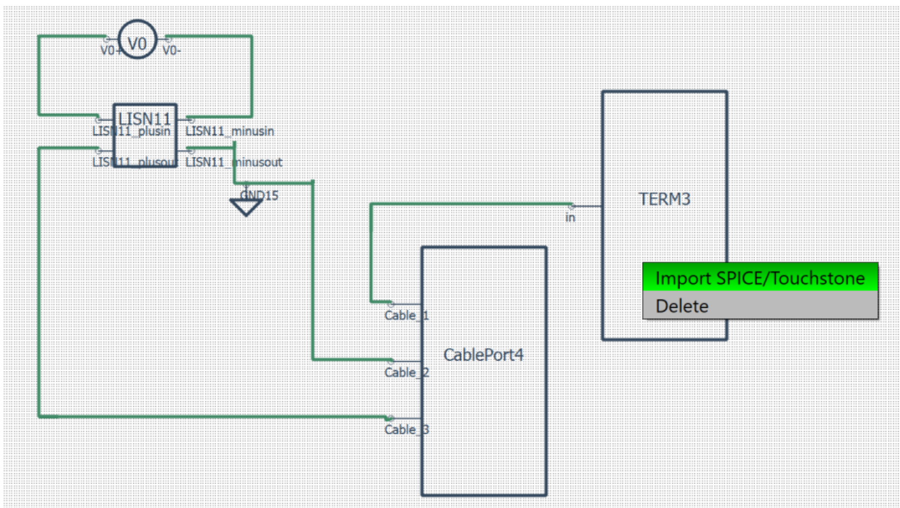


Figure 6-16: Editing the Termination block

Note: If a SPICE file is used, please make sure that there is no "0" or "gnd" or "ground" schematic node in the file. Ground or connect to table must be provided using the "Ground" element. Example of a typical "Termination" file is given below.

```
* demo file
.subckt load cable1 cable2 common
r1 cable1 common 50
r2 cable2 common 50
.ends
```

Figure 6-17: Example Termination file. Note that the "common" node can now be connected to a Ground element using the schematic editor

(e) Drag "Ground" elements if required to the schematic editor.

(f) Make connections by drawing wires between nodes on the schematic editor

6.6 Connector Apparatus

6.6.1 Connector Widget

The connector apparatus/widget can be opened by either of the two methods:

c) Go to "ConnectorDefault" option from the Apparatus List. Double click on ConnectorDefault -> Open Apparatus. The connector widget opens up.

d) Click "Select" button from 3D visualization window and then click on the Connector diagram in the Experiment Main screen.

The figure below shows a connector widget with the related windows and buttons.

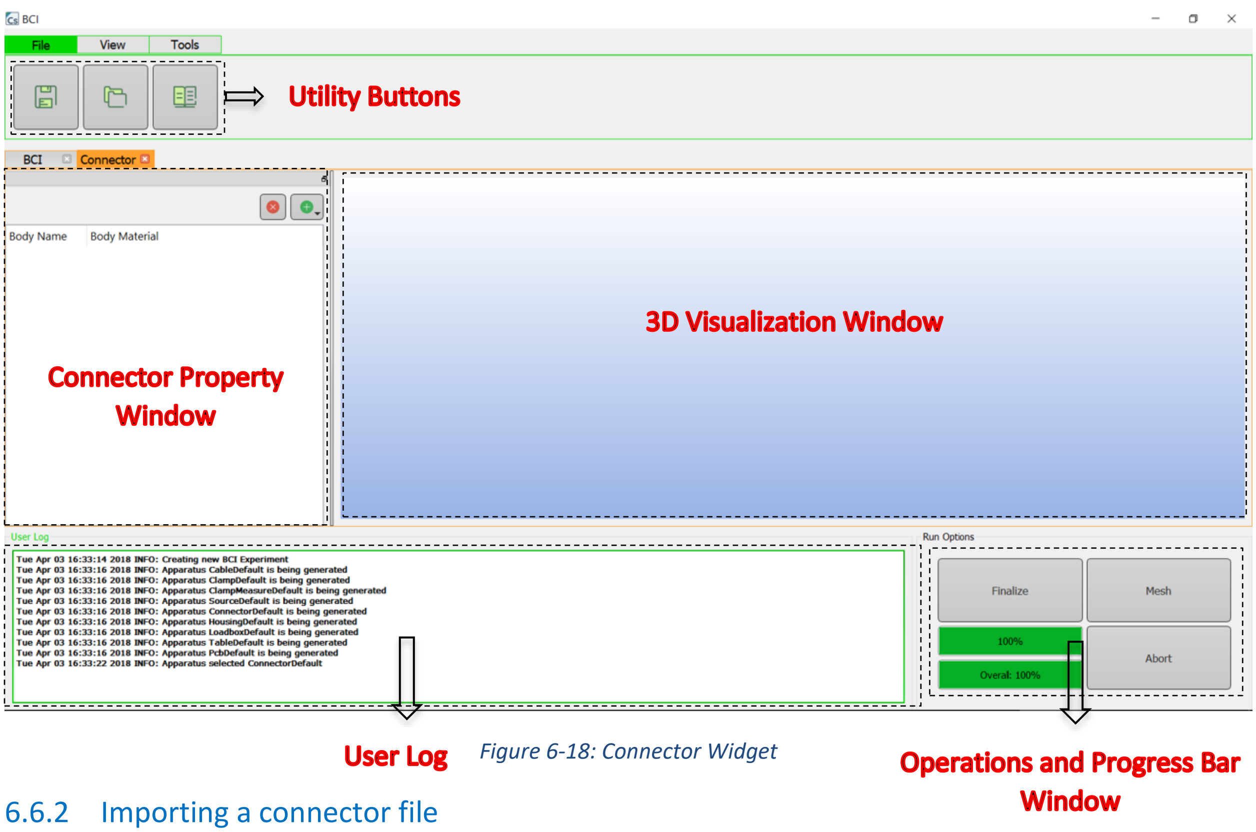


Figure 6-18: Connector Widget

6.6.2 Importing a connector file

The present version of CompScope supports import of connectors in sat (*.sat), step (*.stp and *.step) and IGES (*.igs and *.iges) formats.

To import a connector file, click on load button  from the Utility Buttons panel. Browse to the appropriate file location and import the desired connector file. Once the connector file is imported, it can be viewed in the 3D visualization window. The "view buttons" as described in (cite) can be applied to the connector geometry as needed.

6.6.3 Assigning materials to connector bodies

Once the connector file is imported it can be viewed in the 3D visualization window with all the "View" operations applicable to it. The Connector Property Window on the right of the screen shows different bodies that form the connector along-with a default material (Air). The bodies when expanded also displays the faces which creates them. An example of a connector file imported in the tool and the Connector Property Window populated is shown below:

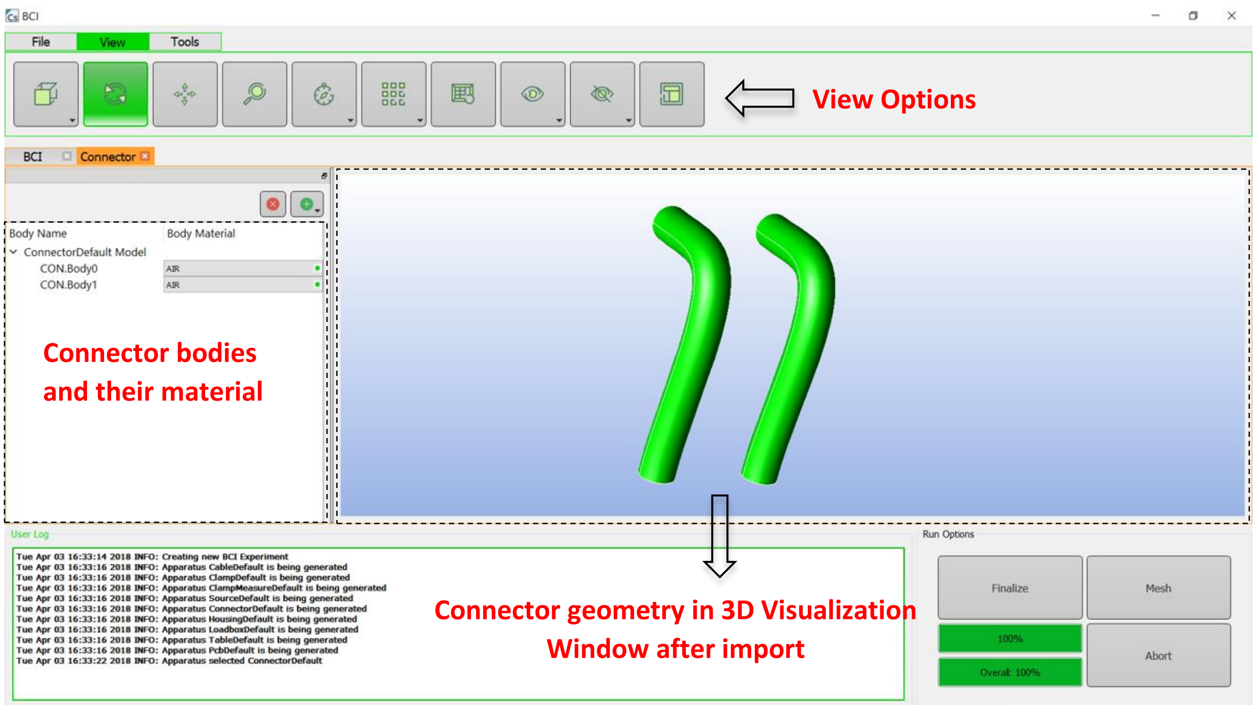


Figure 6-19: Connector Widget populated after importing a file

To assign different materials to different connector bodies:

Step 1: Add the required material in the Material Library Window as explained in chapter 4.

Step 2: Click on the Body Material column beside the appropriate connector body and scroll down to assign the desired material.

Selective viewing: Selective viewing of a connector body is possible by clicking on the appropriate body name. The respective body will be highlighted while the others will be subsequently paled out.

Dielectric viewing: Check the "Show Dielectric" box in View Options to view the dielectric bodies.

The process of assigning materials and different viewing options are shown below:

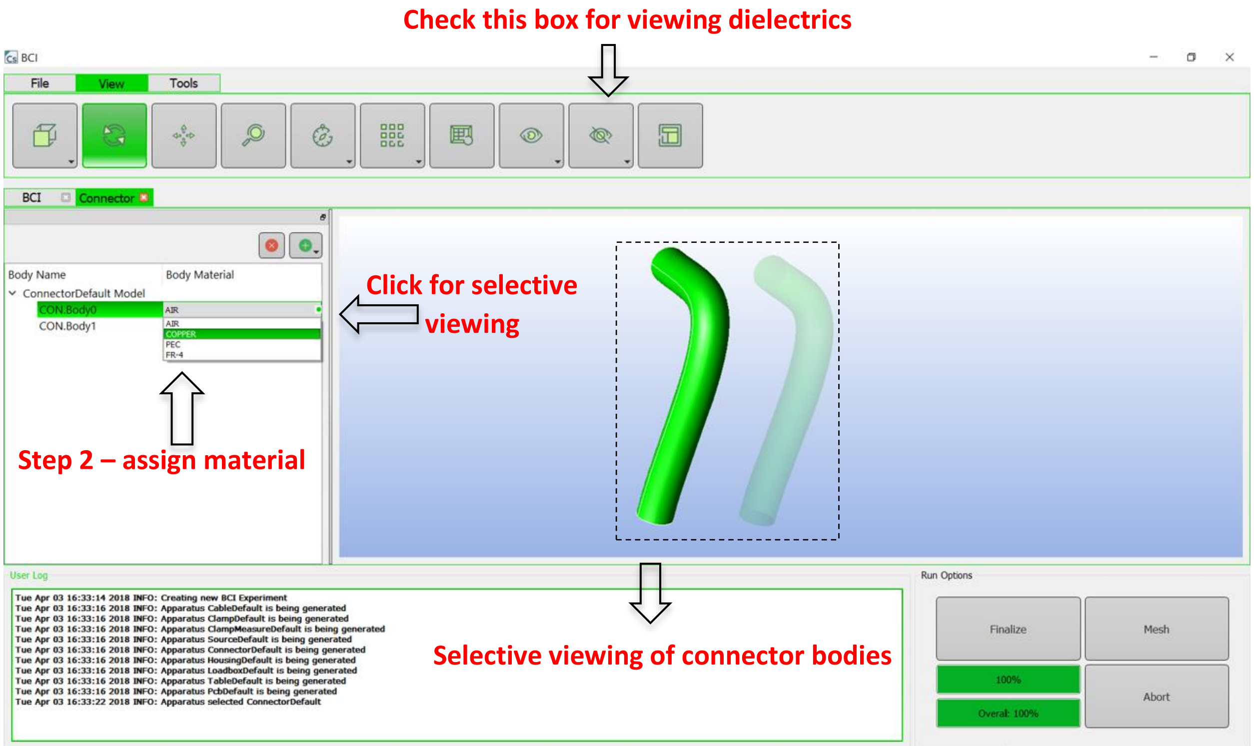


Figure 6-20: Assigning materials and different viewing options

6.6.4 Adding pins on connector bodies

In this section the method to add pins on connector bodies is explained.

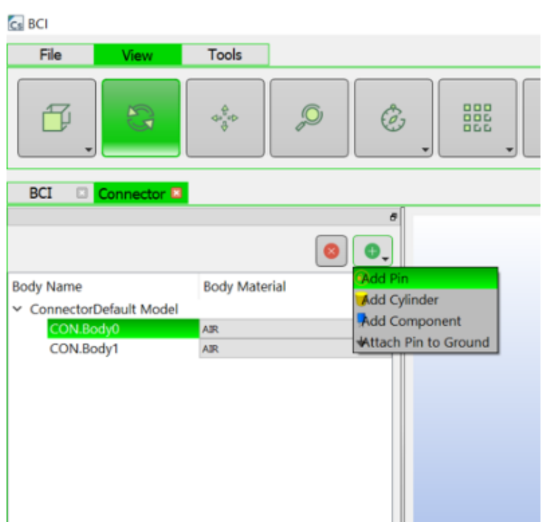


Figure 6-21: Adding pins - Step 1

Step 1: Click on Add button  from the Connector Property Window.

Step 2: Select "Add Pin" option from the list. The "Add Cylinder" option can be chosen to draw a cylinder on the imported geometry. This is more relevant for drawing screws for Housing and is explained in (6.7 Housing Apparatus). The "Attach Pin to Ground" feature is non-functional in this release.

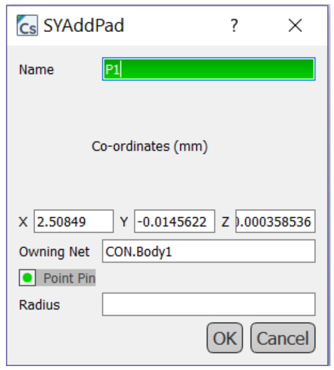


Figure 6-22: Add pin window

Step 3: The "SYAddPad" window opens up as shown in **Error! Reference source not found.**. rovide a name for the pin.

Step 4: Select the "Point Pin" box to designate the pin as a zero-radius pin. Non-point pins are not supported in this release.

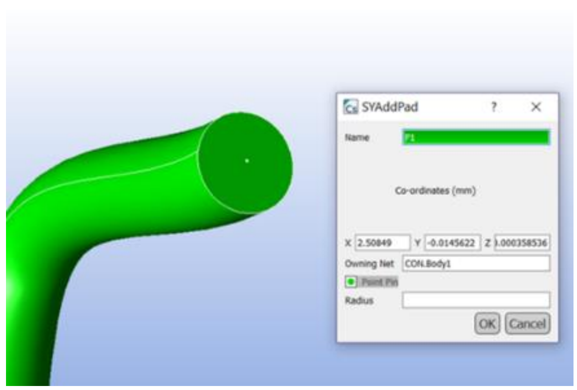


Figure 6-23: Adding pins - step 5

Step 5: Double click on the part of the connector geometry at which the pin is desired to be attached. A white marker appears on the geometry to display the location of the pin. **Error! eference source not found.** illustrates this. The location of the pin can also be entered manually on the "SYAddPad" widget. Click on the "Ok" button to proceed.

Step 6: Repeat Steps 1-5 for adding multiple pins.

The figure below shows the Connector Property Window after a pin has been attached. Please note that the pin details are now added in the window.

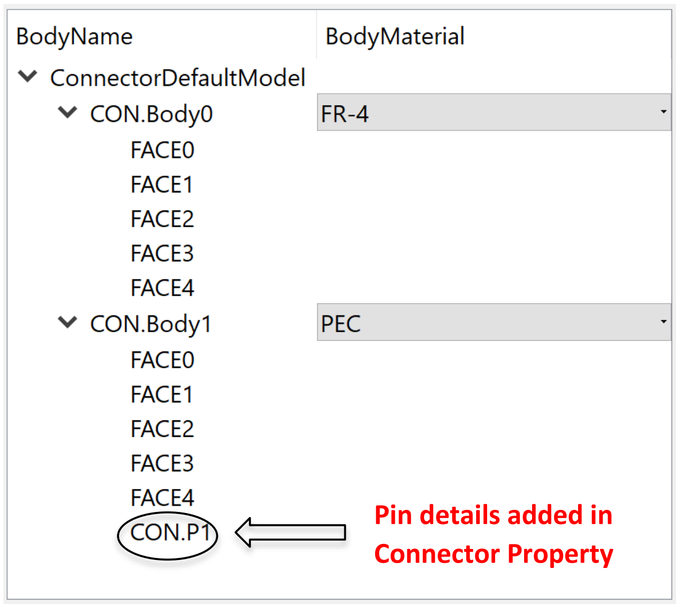


Figure 6-24: Pin details added in Connector Property Window

Deletion of pins: To delete a pin, select the desired pin from Connector Property Window and click on the delete button  .

6.6.5 Operations

Finalize: Once the connector is imported and the materials for connector bodies are assigned, click on "Finalize" button from Operation and Progress Bar Window to finalize the geometry. Material changes are not allowed after finalize operation is completed. However, addition and deletion of pins happens after the finalize operation has occurred.

6.6.6 Video demonstration

The following video demonstrates in detail about importing a connector file, assigning materials, adding pins and finalizing the geometry:

http://simyog.com/resources/manual_animations/Connector.mp4

6.7 Housing Apparatus

All the operations on Housing Apparatus is exactly similar to the Connector Apparatus, and hence the users are suggested to refer to it for further details. Adding cylinder to a housing body is explained in the following video. The cylinders can be assigned materials which are electrically analogous to screws on the housing. Pins can be drawn on those cylinders following similar lines as explained in 6.6.4 Adding pins on connector bodies

(a) Video showing Housing merge to PCB:

http://simyog.com/resources/manual_animations/HousingPCBmergeNoConnector.mp4

(b) Video showing Housing merge to Merged Connector PCB:

http://simyog.com/resources/manual_animations/HousingMergeWithPCBconnector.mp4

6.8 Table Apparatus

BCI: The table dimensions are not considered in BCI simulation. It is assumed to be an infinite conducting plate. The height from table is an important parameter that can be set from 4.3 BCI options.

RE: The table dimensions and height from table are set using the RE options e.g. CISPR 25 parameters.

6.9 Antenna Apparatus (RE only)

Currently, the antenna is not modeled with geometric shape. The antenna position is only used for predicting the electric field values. The position is obtained using the RE options e.g. CISPR 25.

6.10 PCB Apparatus

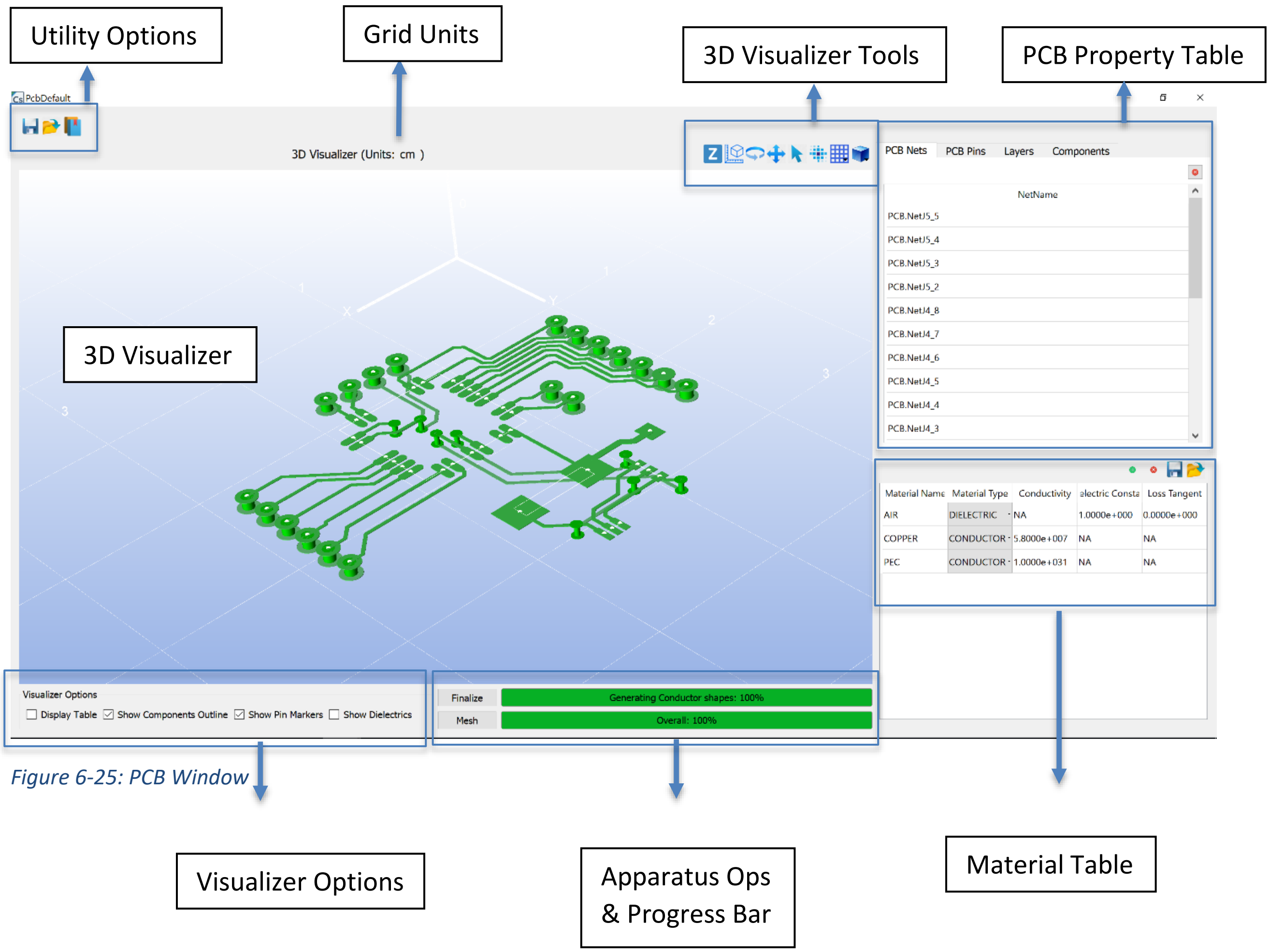
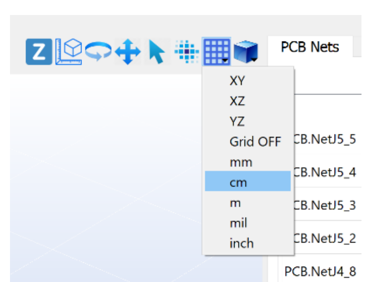


Figure 6-25: PCB Window

6.10.1 PCB Window description

- Utility Options – Refer section 3.2 for usage
  - Load – Supported PCB format ODB++ files
  - Save – User can export 3D model of PCB in STP, OBJ file format
  - Library – Not implemented currently in CompScope.
- Visualizer Tools – Refer section 3.2 for Orbit, Pan, Select, Mesh View, Projection. Grid View, Measurement Tool and Z-axis Zoom View are 3 other tools which provided by PCB Apparatus Window
  - Grid – CompScope supports Grid Insertion with different units. User can select three different styles in defined by plane of grid i.e. XY, YZ, XZ grid. Figure



6-25 shows XY grid. To turn off grid select “Grid OFF” option.

- Measurement tool
  As name suggest it can be used to measure distance between two arbitrary points on Apparatus. Distance measured is displayed in mm. To measure distance, click on Measure Tool option and then select two points by double clicking at required positions of the Apparatus While measuring, mouse left button single click-hold and move can be used to orbit the PCB and middle mouse button click-hold and move can be used Pan around. Refer Figure 6-27(a): Measurement Tool.

- Z-axis zoom view
  This is a special tool just applicable for PCB Apparatus. PCB being layered structure typically with tiny layer widths compared to its size, it could be necessary to provide an artificial expansion to Z-axis in order to view different layers clearly and identify Via between them. Z-axis zoom tool helps to do this. Click on “Z”- button to select tool. Press and hold **Right mouse** button while dragging mouse to adjust z-axis zoom. While zooming z-axis, mouse left button single click-hold and move can be used to orbit the PCB and middle mouse button click-hold and move can be used Pan around. Selection of any other tool (orbit, pan etc) will remove Z-axis zoom and bring back to original state. Refer Figure 6-26(b): PCB after Z-axis zoom.

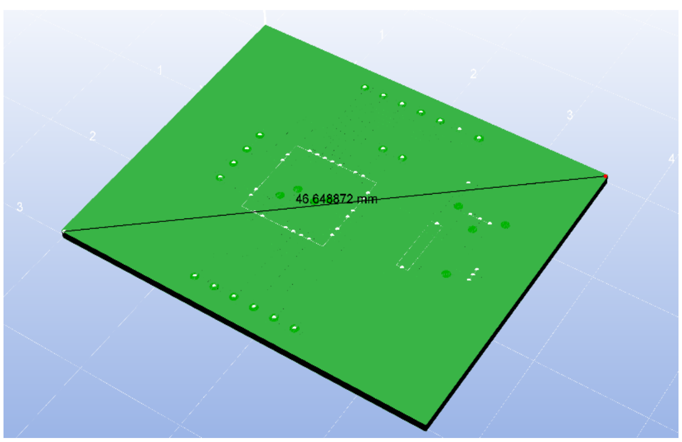


Figure 6-27(a): Measurement Tool

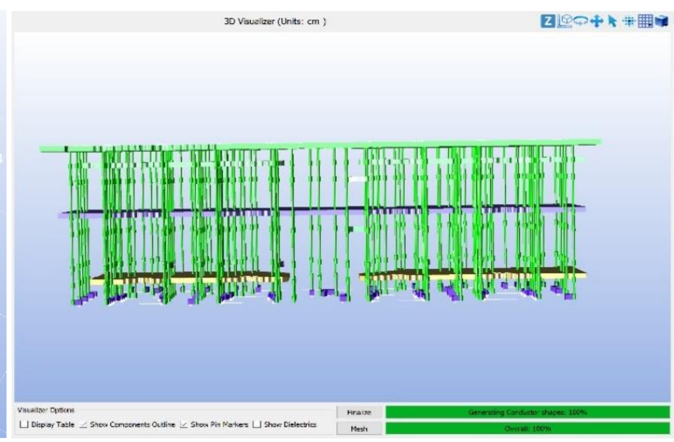


Figure 6-26(b): PCB after Z-axis zoom

- Grid Units
  - On selection of particular unit under Grid option Units box reflects corresponding grid unit. For example in Figure 6-25 grid unit is “cm”.
- PCB Property Table

- CompScope handles PCB as construction of four basic properties – nets, pins, layers and components. Each of these are displayed in a table. These property tables are discussed in detail in sections 0 to 0.

- Material Table

  Refer Chapter 4 for material handling.
- Apparatus Operations and Progress Bar

  Progress Bar displays progress of PCB import and operations. Refer 9.1 and 9.2 for apparatus operations.
- 3D Visualizer
  - Displays imported PCB. It supports highlighting of nets, pins, components and layer-wise visualization of PCB.
- Visualizer Options
  - Display Table – Deprecated, should not be used.
  - Show components outline – Shows a white bounding rectangle around component when checked. Can be turned off by unchecking this box. Refer Figure 6-28.
  - Show Pin Markers – PCB pins are denoted bright white points on PCB when checkbox is checked. Can be turned off by unchecking this box. Refer Figure 6-29.
  - Show Dielectrics – PCB dielectric layers by default are invisible as checkbox is in unchecked position. All dielectrics can be turned on by checking the box. Refer Figure 6-30.

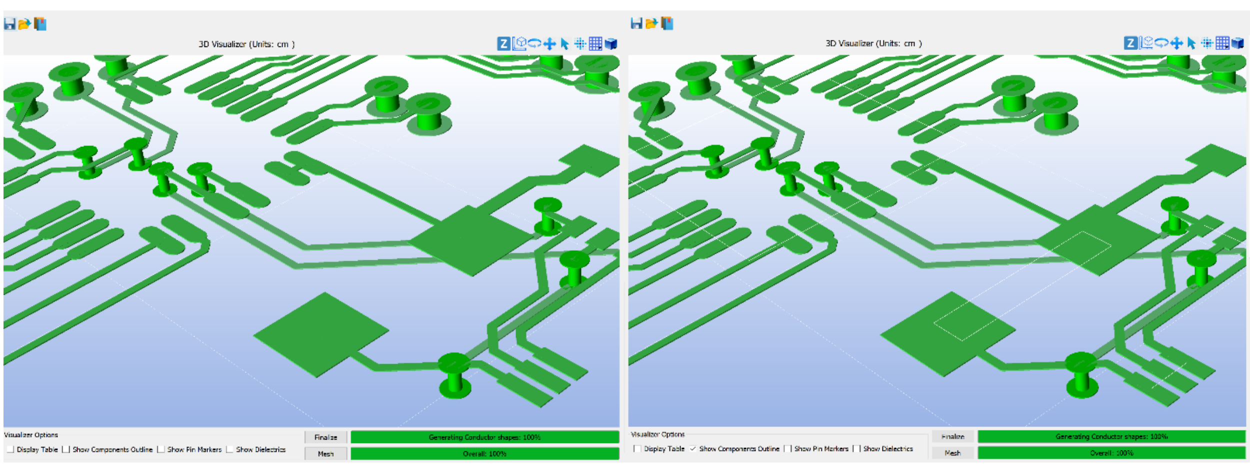


Figure 6-28: Component Outline Showing/ Hiding

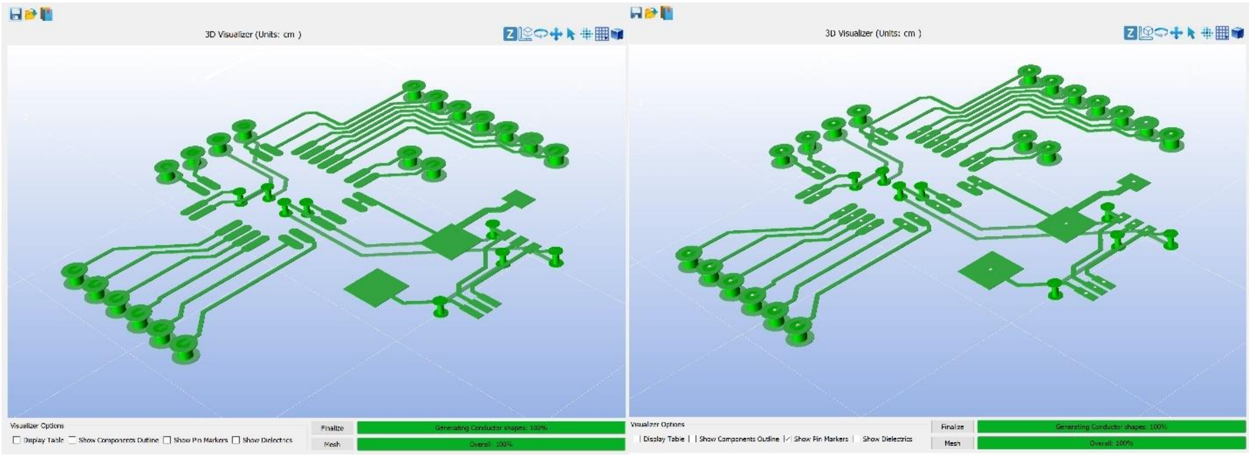


Figure 6-29: PCB pins showing/ Hiding

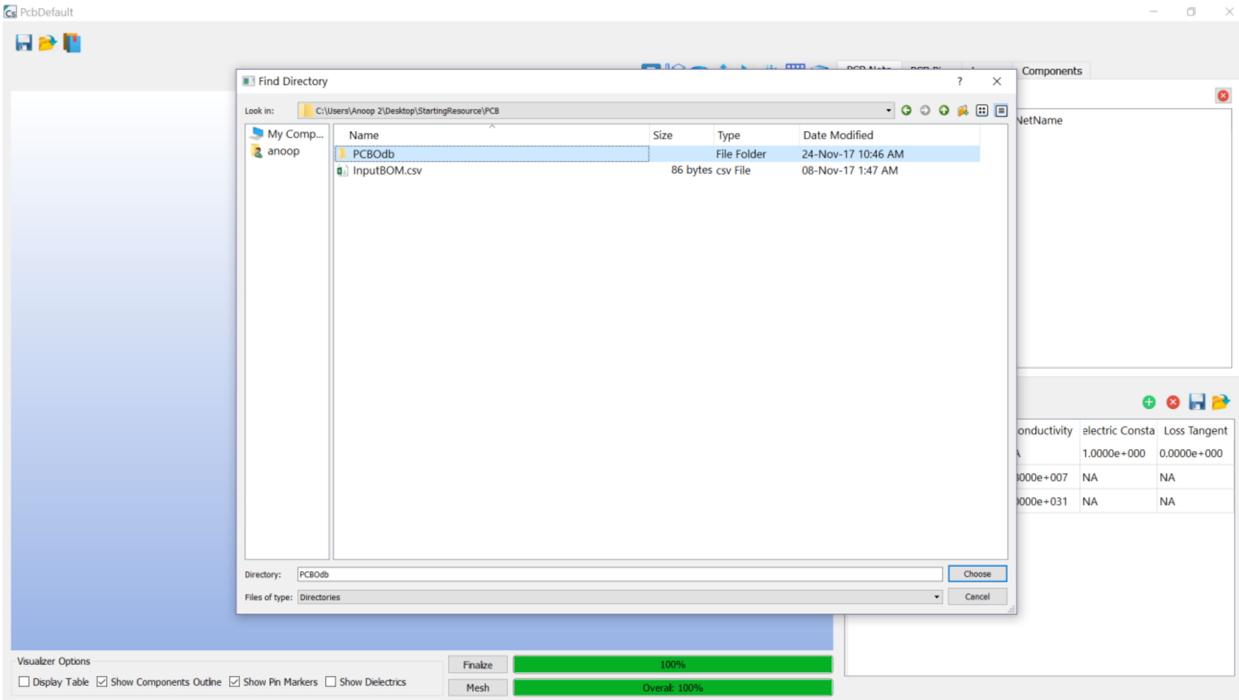
Figure 6-30: Dielectric layers Showing/ Hiding

6.10.2 Import ODB++

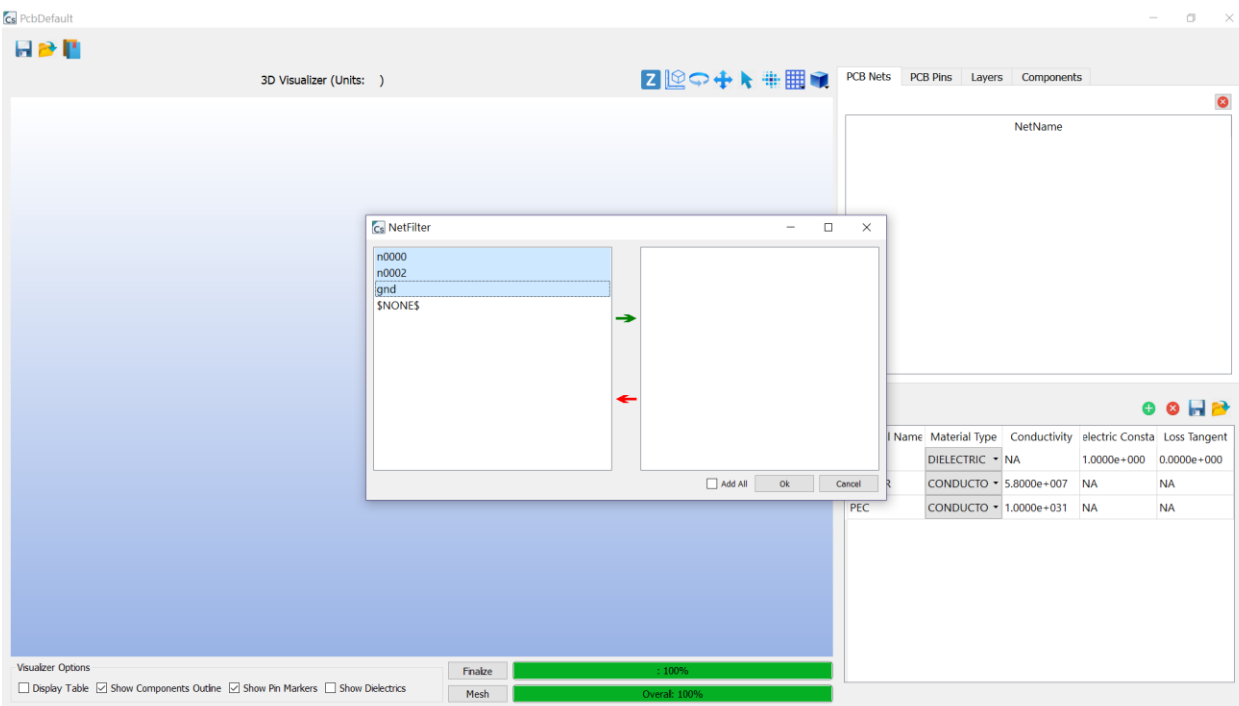
Step 1: Click on the 'Load' button

Step 2: Directory browse window will open up, browse to the location of the ODB++ file, select it and click on choose.

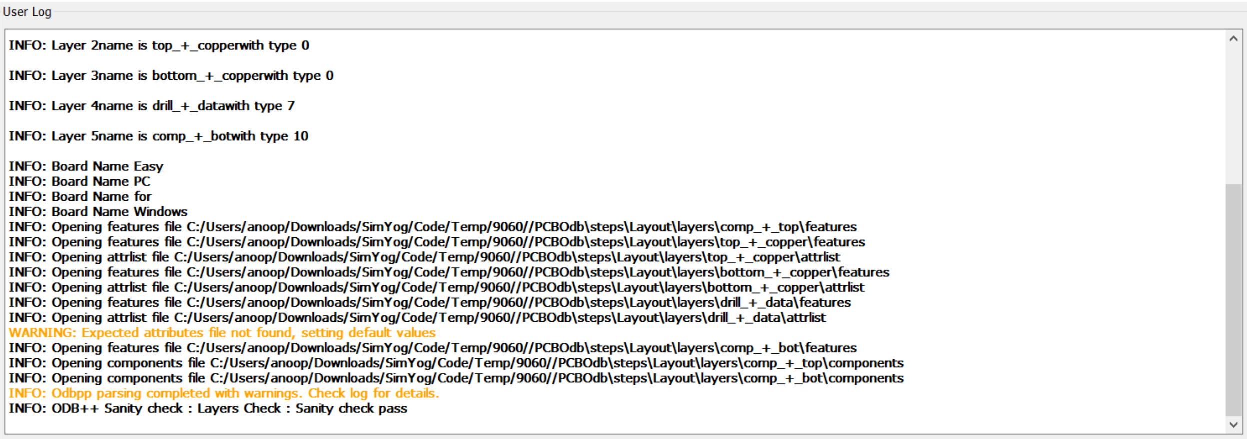
Note that ODB++ files are accepted in zip, tgz and folder format.



Step 3: Net Filter window will pop up, select the desired nets and add to selected net list and press 'OK'. If you want all nets to added, check "Add All" and continue.



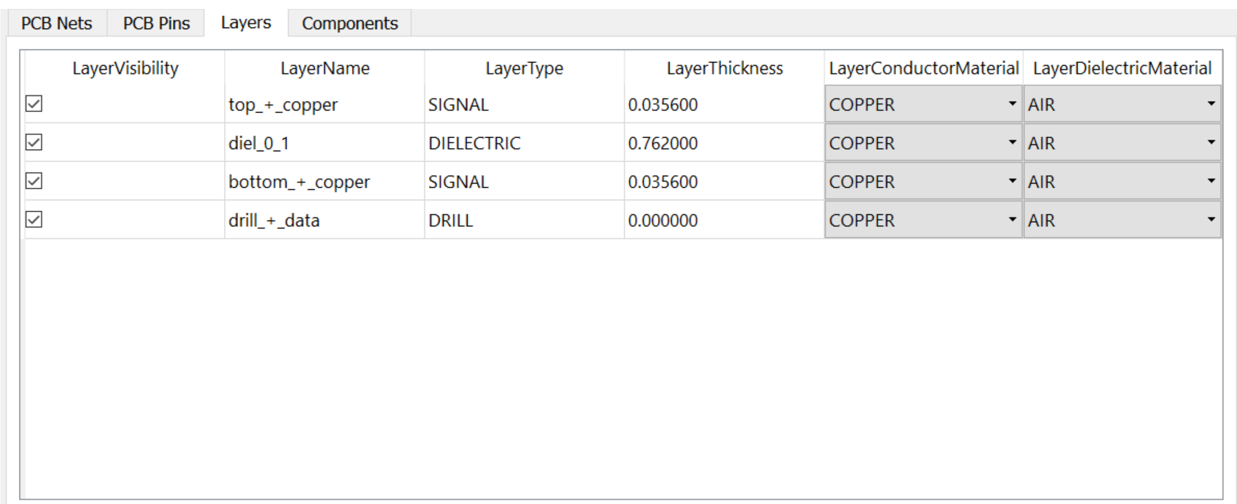
Step 4: Log messages window displays the info and warnings that occurred during ODB++ import



Step 5: Please wait for a moment till rendered PCB appears on 3D Visualizer and Property tables get populated.

6.10.3 Stackup

The Stackup can be accessed by clicking on the 'Layers' tab of the 'PCB Property Window Manager'.



LayerVisibility	LayerName	LayerType	LayerThickness	LayerConductorMaterial	LayerDielectricMaterial
☑	top_+_copper	SIGNAL	0.035600	COPPER	AIR
☑	diel_0_1	DIELECTRIC	0.762000	COPPER	AIR
☑	bottom_+_copper	SIGNAL	0.035600	COPPER	AIR
☑	drill_+_data	DRILL	0.000000	COPPER	AIR

NOTE: Stackup is shown in order of decreasing elevation, that is bottom layer at the last and top layer is first.

NOTE: A Drill layer might be present and it is a part of the ODB++ stackup hierarchy and it serves no purpose for simulation.

NOTE: Changing Layer names in not supported in current Compscope version.

Changing layer thickness: Layer thickness of all the layers can be done, however the model is only updated upon finalize.

Changing Material: user can select desired material for layer dielectric and conductors from drop down present under LayerDielectricMaterial, LayerConductorMaterial columns. Conductor and Dielectric Materials present in the material window can be assigned to stackup. By default AIR, COPPER and PEC exist, user can add material from library or can create custom material as well.

Layer Visibility: Before the finalize operation is performed individual layers can be hidden or showed using the 'LayerVisibility' checkbox. On toggling layer visibility checkbox perform single click anywhere in 3D Visualizer to update the PCB view. Note that LayerVisibility does not affect the physical model. Currently CompScope does not support layer deletion or new layer addition.

6.10.4 Nets

Net Table under PCB Properties lists PCB nets imported by ODB++ load.

Nets support forward and backward highlighting. Forward highlighting is selecting net from table highlights corresponding 3D view of the net making other nets translucent. Backward highlighting uses Selection Tool to select particular 3D object in PCB view. On selection corresponding net in the Net Table will get selected, it also highlights particular part of net where selection was performed with different color than highlight color of full net.

After importing if user doesn't wish to keep some nets, net deletion can be used. Select a Net in Table and click on Delete Button at top right of the Net Table. Please note deleted nets cleared from visualization only after you 'Finalize' PCB. Refer 9.1 for Finalize operation. If user had already clicked Finalize before deletion, they need to perform it again after deletion. Currently CompScope doesn't support addition of user defined Nets.

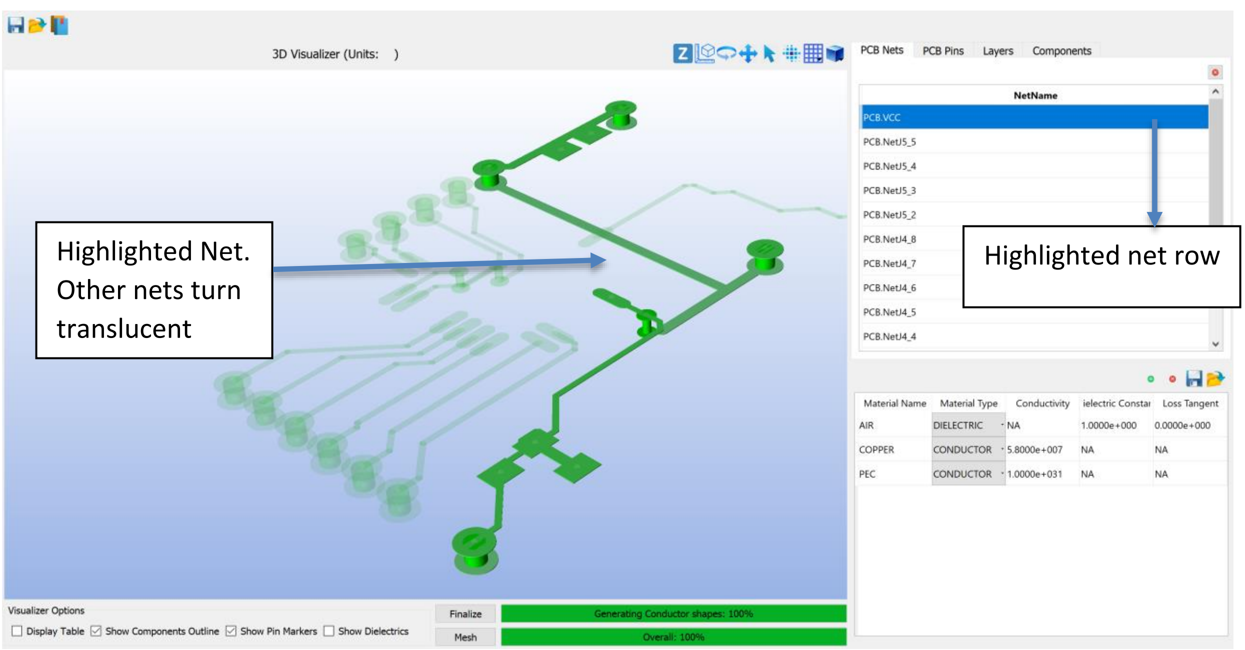


Figure 6-31: Net Highlighting (Forward)

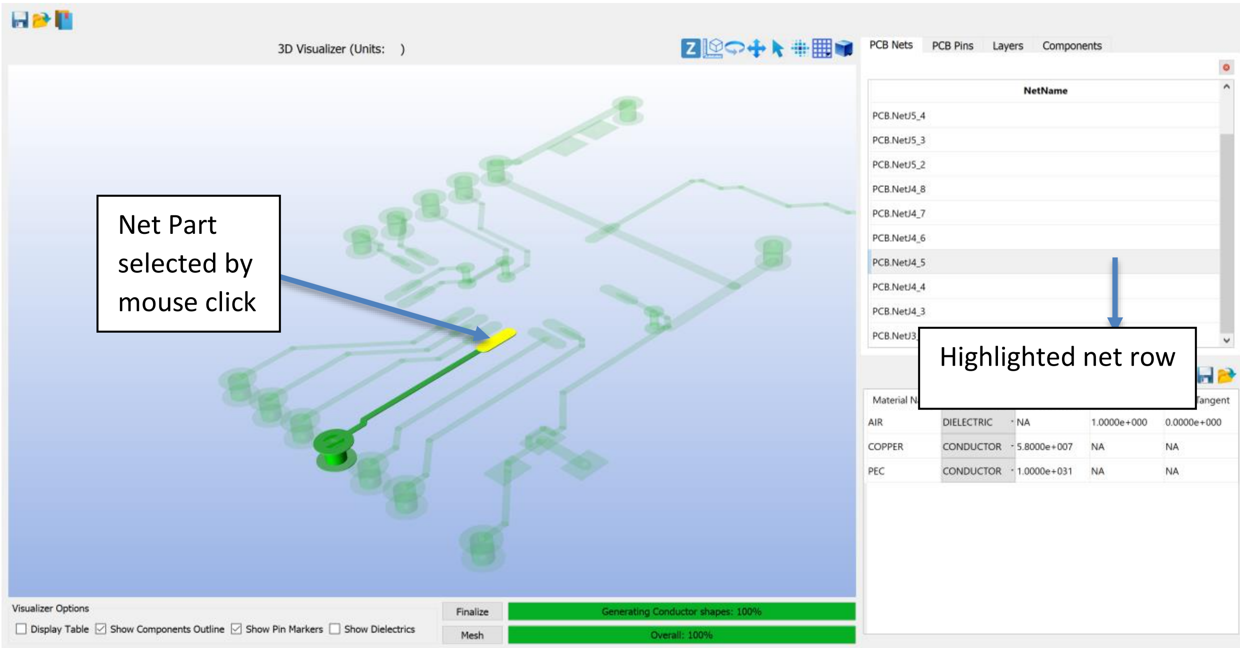


Figure 6-32: Net Highlighting (Backward)

6.10.5 Pins

Pins of a PCB are virtual points where components can be connected. PCB Pins are also used for connecting PCB to other apparatus like cable, connector and housing. By default ODB++ import will create many pins on PCB where components can be connected or components already exist. Similar to nets pins can be highlighted in forward and reverse way.

At right top corner of Pin Table there is “Ground Option”. Using this option user can connect particular pin of PCB to Ground. Current version of CompScope does not support this operation and user should not use it.

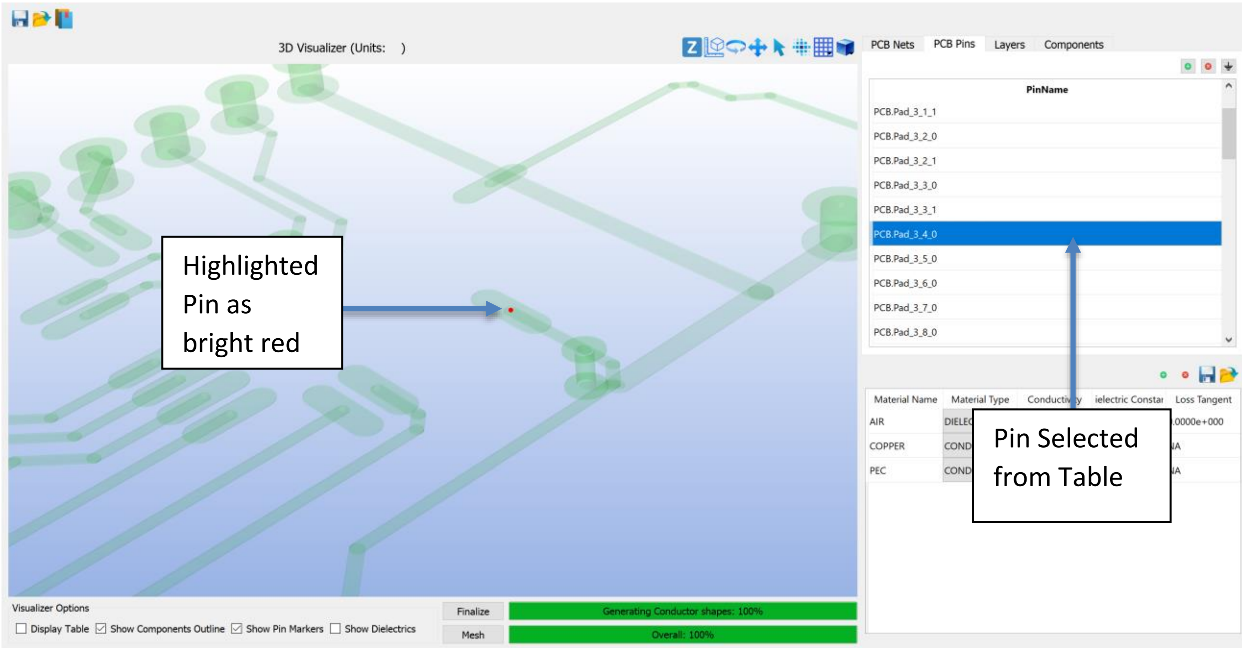


Figure 6-33: Pin Highlighting (Forward)

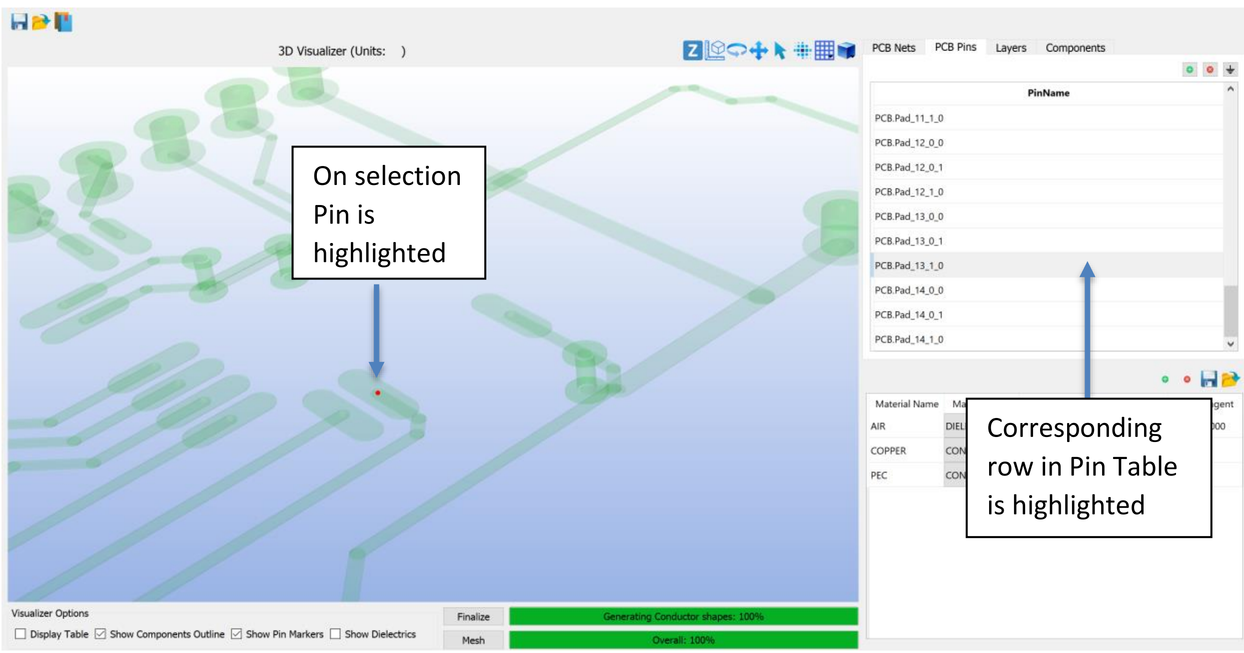


Figure 6-34: Pin highlighting (Backward)

User can also create new Pins and delete existing.

- To delete Pin select pin you want to delete from Pin Table and click on Delete button at right top corner of the table.

- To add a new Pin follow below steps -

  Step 1: Click on Add Button at right top corner. It will pop up a New Pin Addition Dialog.

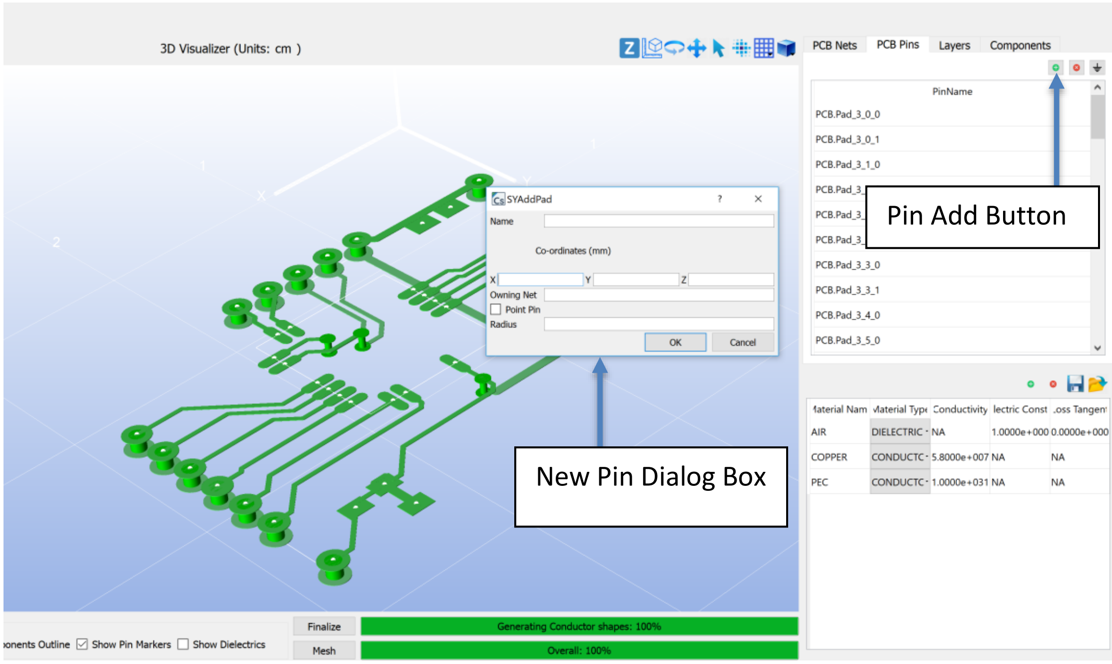


Figure 6-35: Pin creation step 2

Step 2: Double click on required location of PCB, it will automatically update pin co-ordinates and owning net of Pin (Net on which pin has been added) in New Pin Dialog.

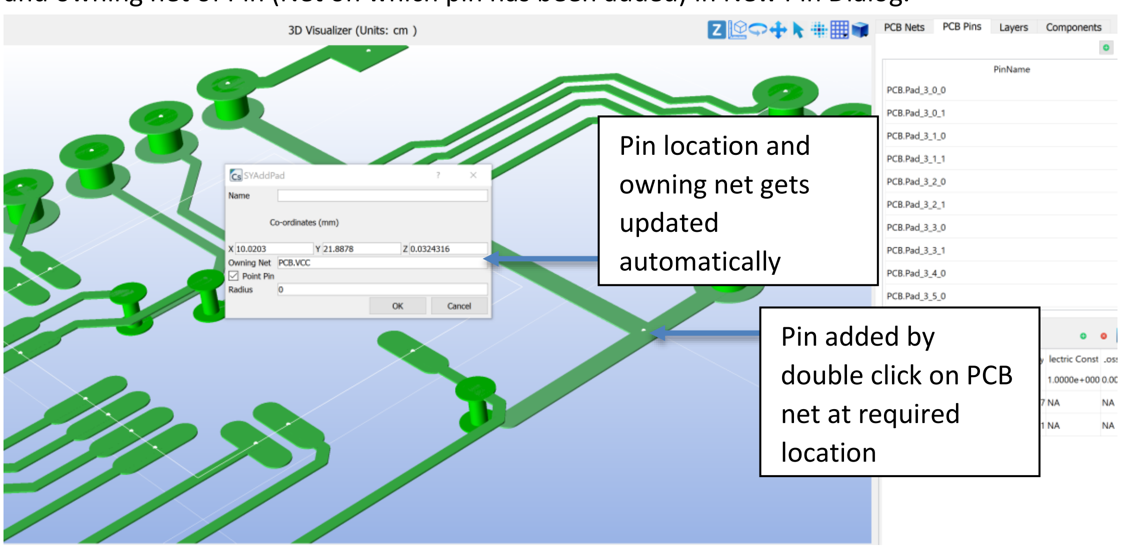


Figure 6-36: Pin creation step 2

Step 3: Give name to pin and click OK. Make sure that "Point pin" is always checked as current version doesn't support other than point pins. Once Pin gets added successfully, Pin Table gets updated to show user created pin at the bottom of table.

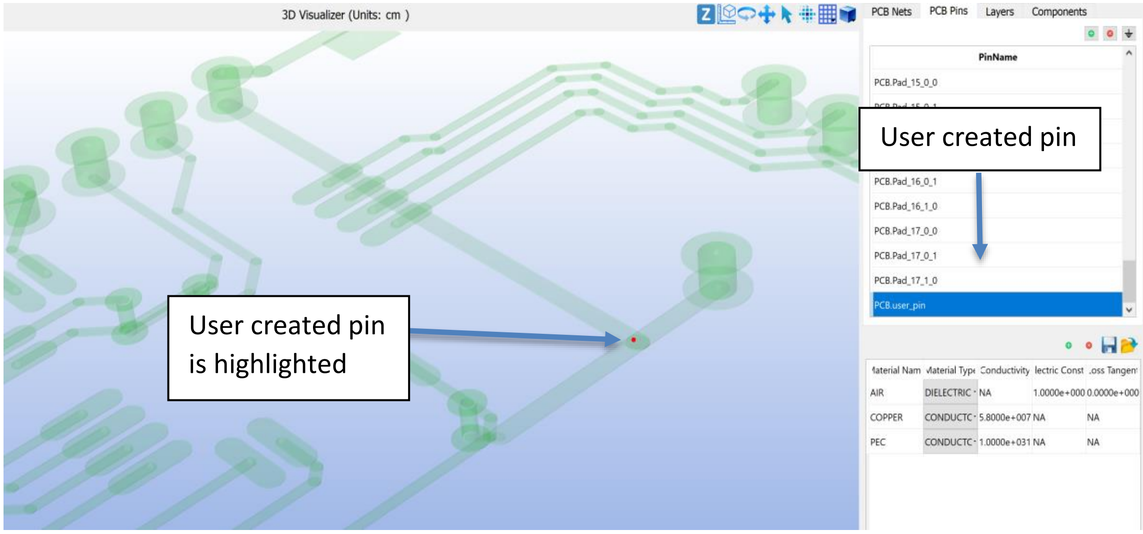


Figure 6-37: Pin creation step 3

6.10.6 Components

Component Table populates existing components in ODB++ on its import. It also supports new component creation, editing existing component, deletion and highlighting. CompScope supports different types of components – simple, SPICE and IC model.

Component has two important properties ‘Enable’ and ‘Probe’. ‘isEnabled’ field in the table when checked decides whether component is considered in simulation. ‘isProbed’ causes CompScope to compute and store voltage/current across/through the component. Default ODB++ components (which are part of designed PCB) are imported as empty components, thus show Component Type as ‘Unknown’ and not Enabled, not Probed. To enable these components and supply values in a go user has to import Bill Of Materials (BOM), this method is discussed in 6.10.6.6. To enable these default components user can as well edit them one-by-one. Editing of components is discussed in 6.10.6.2.2 and 0.

6.10.6.1 Component Highlighting

Only forward highlighting is supported. On selecting component row from Table Component outline is highlighted in Visualizer.

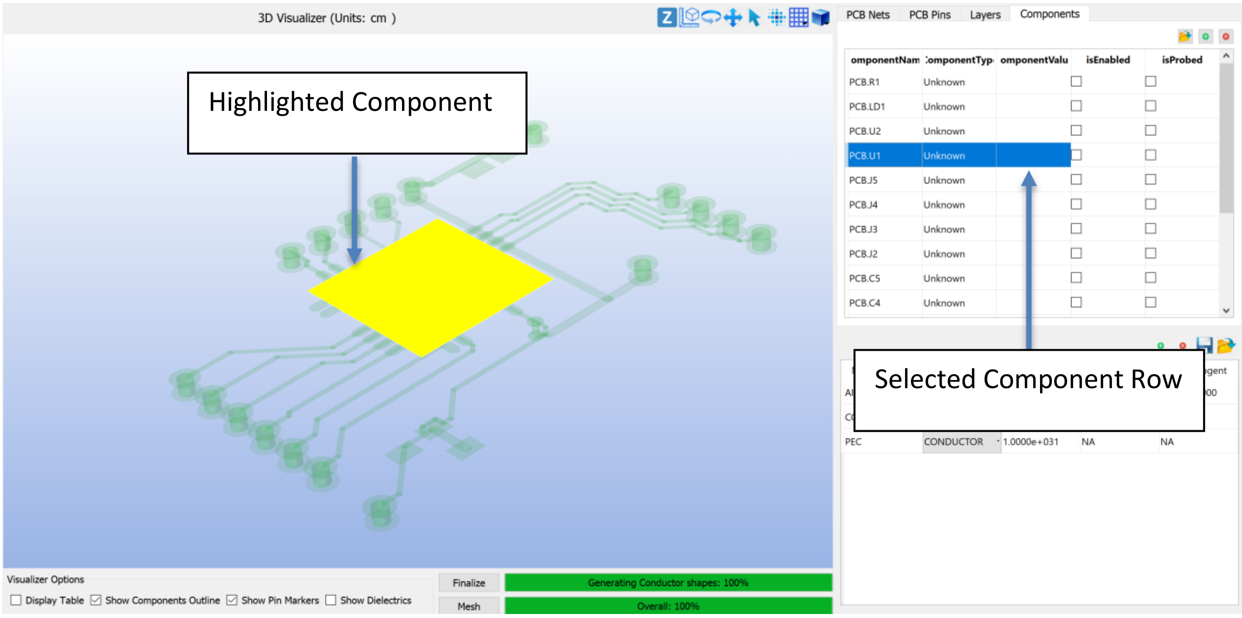


Figure 6-38: Component Highlighting (Forward)

6.10.6.2 Simple Component

Simple component includes Resistor, Inductor and Capacitor. Capacitor and Inductors support parasitic values like ESR and EPC/ESL.

6.10.6.2.1 Adding a new Simple Component

Step 1: Click on Add Component at top right corner of Component Table. It pops up a two-step Dialog for new component. Select required pins in between which the component will be connected. As shown in Figure 6-39 when two pins for a simple component are selected PCB Visualizer highlights those pins. Click Next after pins are selected.

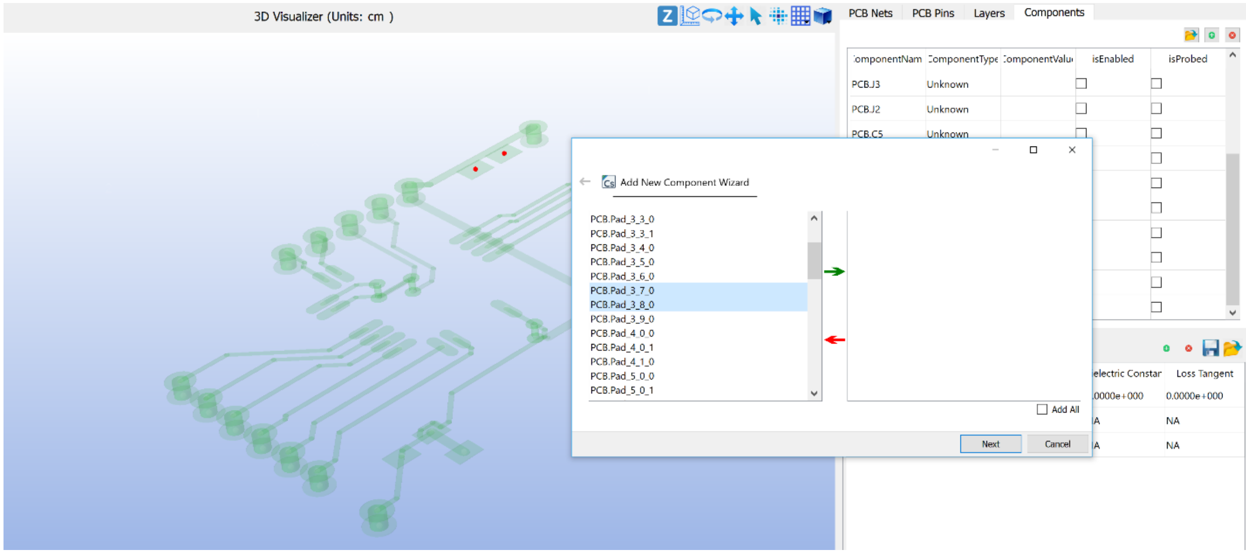


Figure 6-39: New Component Addition Step-1

Step 2: In next step of Add Component Dialog selected pins are shown in column on left and fields to populate values and name of simple component are shown in box at right. Put appropriate values and name for component. Select component type out of Resistor, Capacitor, Inductor. For Resistor type ESL/EPC and ESR are ignored. For Capacitor with ESR and ESL appears automatically while for Inductor EPC appears instead of ESL. Refer Figure 6-40.

“Set As Probe” check box when checked, causes CompScope to compute and store voltage/current across/through the component. If user wants monitor voltage or current for the component it is required to check this box. Once experiment is solved results of these ‘probes’ will be available at Results Window. Refer Chapter 10 for detailed discussion of results plotting.

Step 3: Hit on Finish once values are entered in all input boxes. A new component gets created and added at the end of Component Table as shown in Figure 6-41. If you note component type is set to Resistor and it is ‘Probed’ as we had checked the “Set As Probe” while creating it. Also its “isEnabled” field is checked by default. In case user does not want to use the component, they can edit this to not enabled and CompScope’s solver won’t consider it while simulation.

6.10.6.2.2 Editing a Simple Component

To edit any simple component double click left mouse button on the corresponding Component Table row, it will pop-up a dialog similar to Figure 6-40 with existing component value, type etc. User can modify these and click Ok button to edit the component.

Note – Editing of component pins and name is not supported in current version of Compscope.

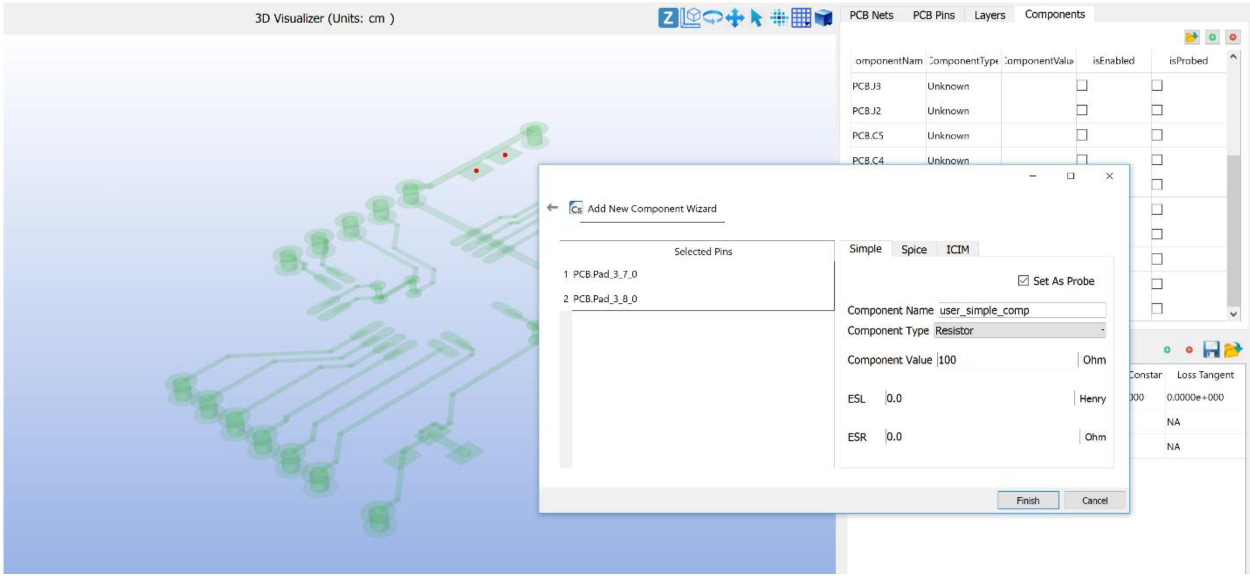


Figure 6-40: New Component Addition Step-2

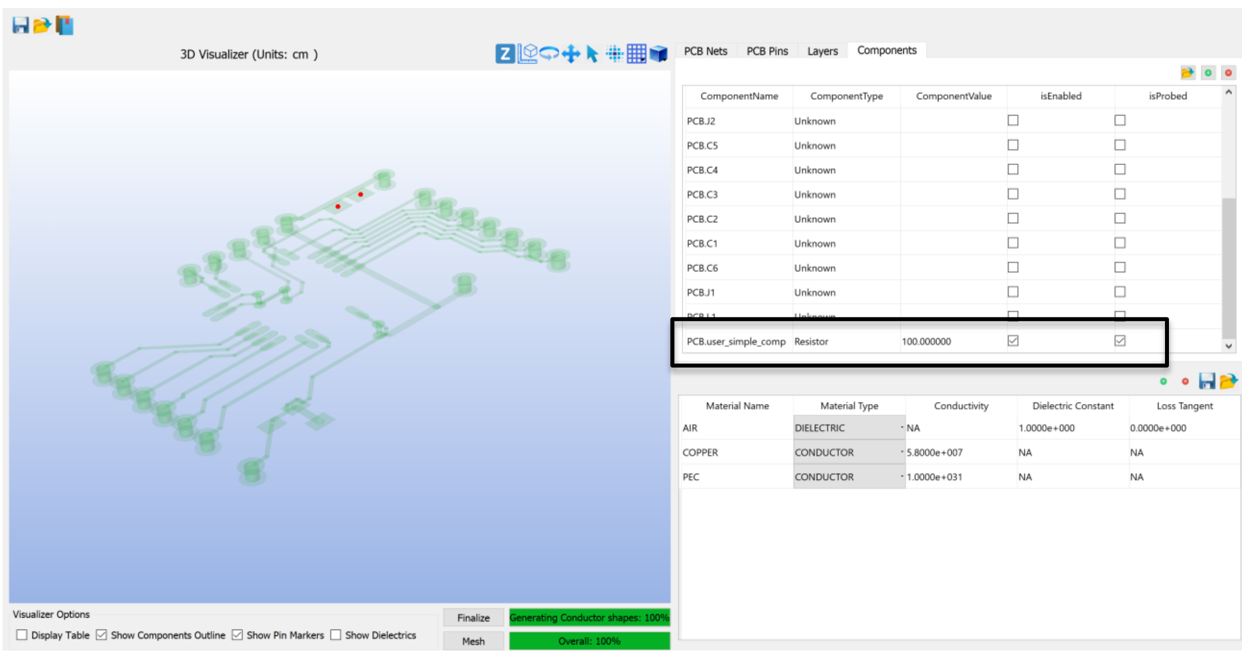


Figure 6-41: New Component Addition Step-3

6.10.6.3 SPICE and Nport

CompScope supports importing multiport block component, both in spice or touchstone format. In case of touchstone block user needs to map the common reference port to one of the selected pin along with the ports. Similarly, spice sub-circuit pins can be mapped to the selected pins including internal ground pin of the sub-circuit. Check below subsections for detailed steps of creating a new component as Spice/Nport data.

6.10.6.3.1 Adding a component as SPICE model

Step 1: Let's assume that SPICE sub-circuit we want import has 4 nodes including internal GND node. To map these nodes select four pins on PCB in step 1 of New Component Dialog. As shown in Figure 6-42 these pins get highlighted on selection. Hit Next button once required pins are selected.

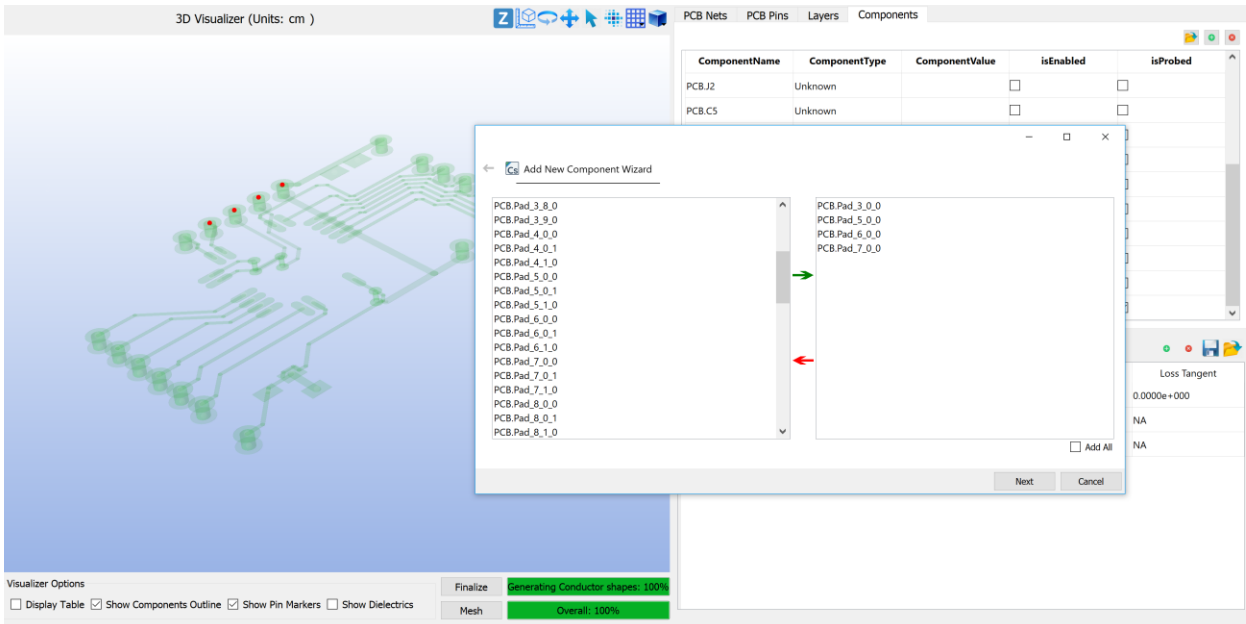


Figure 6-42: Import SPICE circuit as Component (Step 1)

Step 2: Browse for the Spice file. As soon as CompScope receives a valid sub-circuit file nodes of the spice block are populated in drop-down menu under the column 'Subcircuit Pins' as shown in Figure 6-43

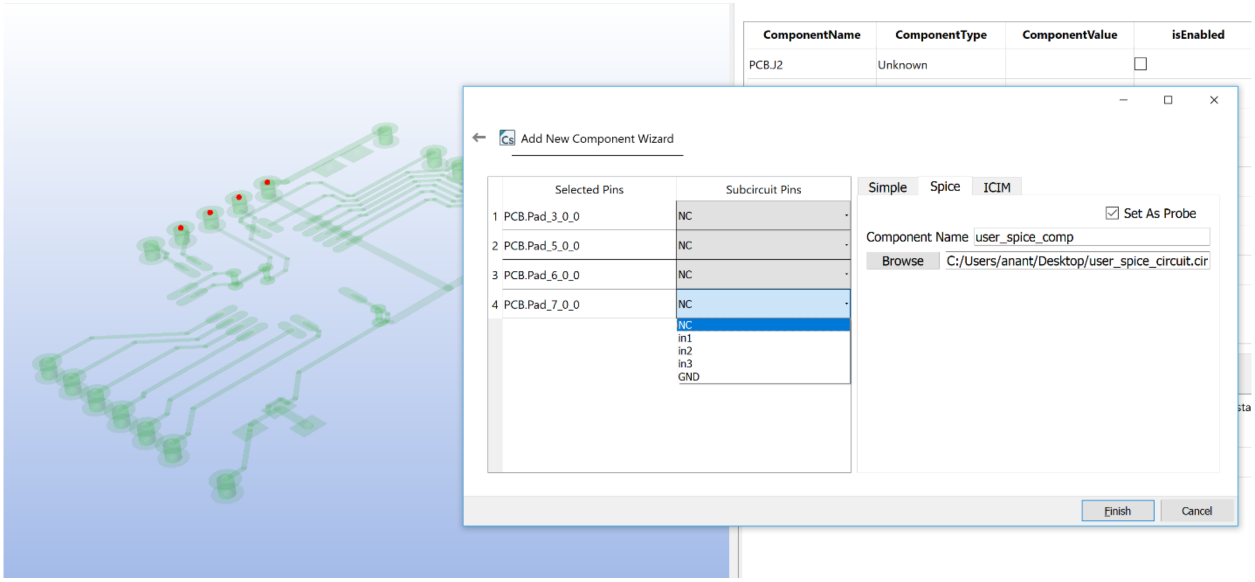


Figure 6-43: Import SPICE circuit as Component (Step 2)

Step 3: Assign sub-circuit nodes to PCB as per required connectivity (Figure 6-44). Give component name and probe setting. Press Finish to add new SPICE circuit component.

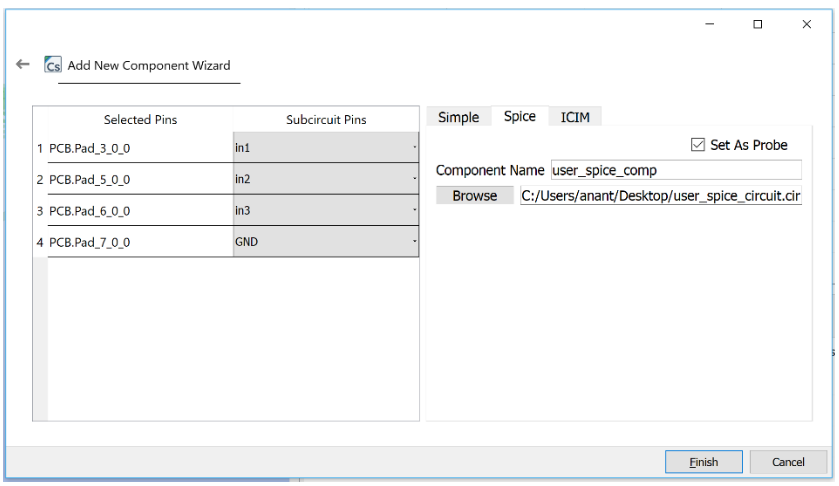


Figure 6-44: Spice Node to PCB pin mapping

Added new component shows type as 'SpiceSubckt'.

ComponentName	ComponentType	ComponentValue	isEnabled	isProbed
PCB.user_spice_comp	SpiceSubckt	0.0	☑	☑

6.10.6.3.2 Adding a component as Nport Data

Similar to adding component as Spice file, browse touchstone file (.s*n*p where *n* is number of ports). Drop-down menu under subcircuit-pins automatically detects touchstone file ports and populates them. User can assign PCB pin to Touchstone file ports with one pin to Common Reference. Give component name and probe setting. Press Finish to add new Nport component. Added component type will show up as ‘NportData’.

ComponentName	ComponentType	ComponentValue	isEnabled	isProbed
PCB.user_nport_comp	NportData	0.0	☑	☑

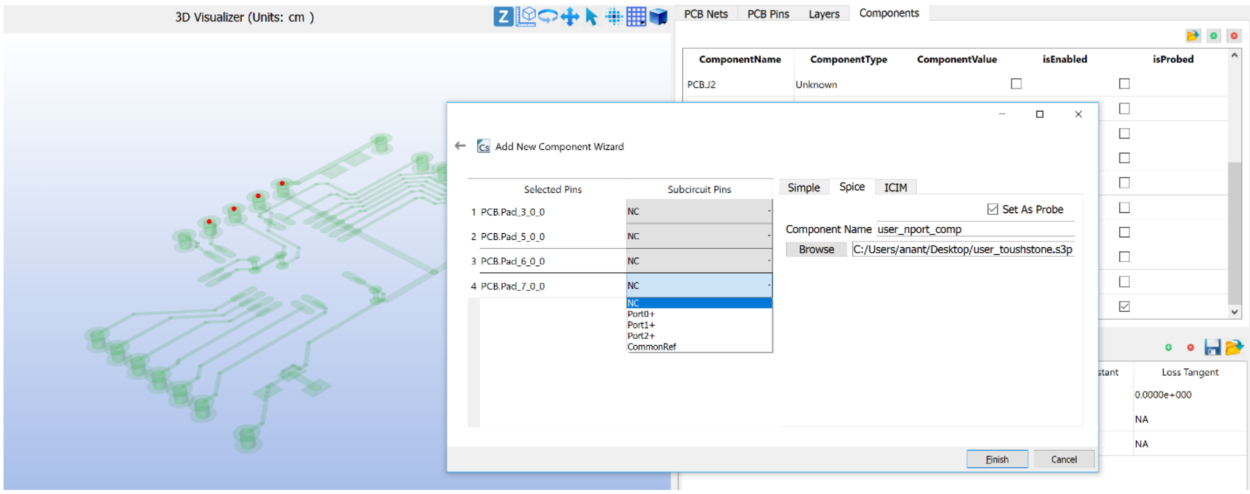


Figure 6-45: Add component as Nport data

6.10.6.3.3 Editing SPICE/Nport component

Editing of SPICE sub-circuit or Nport data component is similar to Simple component and triggered by double click on component table row. User can browse different file reassign pins to nodes or ports on modify the component properties.

6.10.6.4 IC Model

Refer to Chapter 7 for detailed IC model discussion.

6.10.6.5 IBIS

Input/Output Buffer specification abbreviated as IBIS are behavioral models describing the IC. The reader can refer to the IBIS manual version 6.1 which is currently the latest supported version. The following paragraph explains how to create an IBIS component in *CompScope*.

Step 1: It is essential that the user is aware of the required number of pins to create the IBIS component, since the number of pins in a component cannot be edited once created. To create the IBIS component, click on the “Add Component” button and select the desired pins.

Figure 6-39: Add component as IBIS data (Step 1)

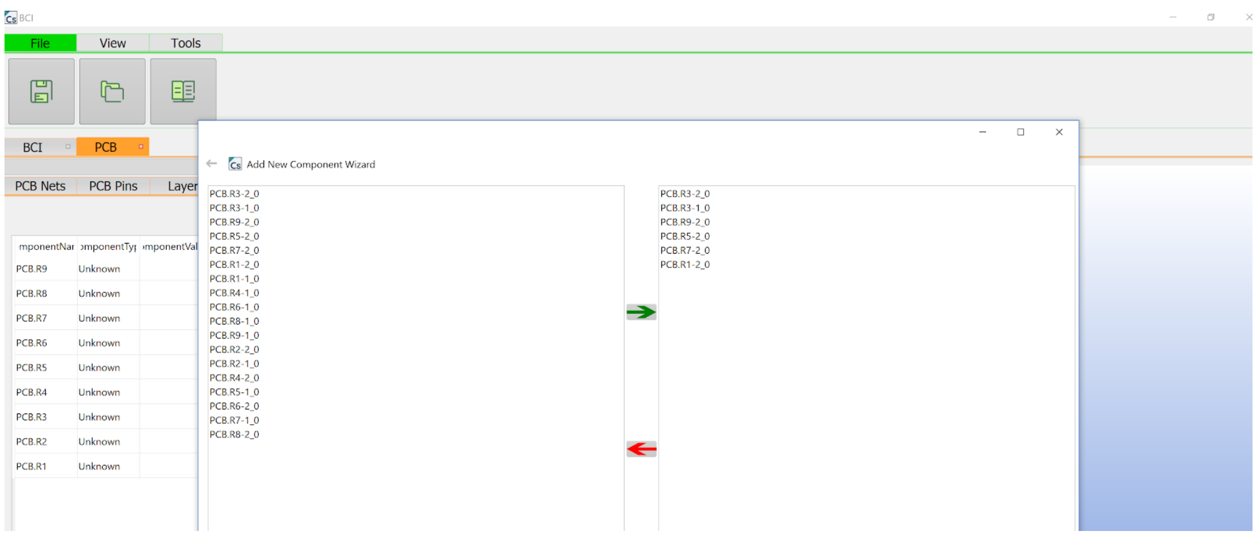


Figure 6-40: Select the Desired Pins (Step 1)

Step 2: Navigate to the IBIS Tab and enter the name of the component to be created and click on browse. In the *File Browser* which opens, select the desired *.ibs file to be imported.

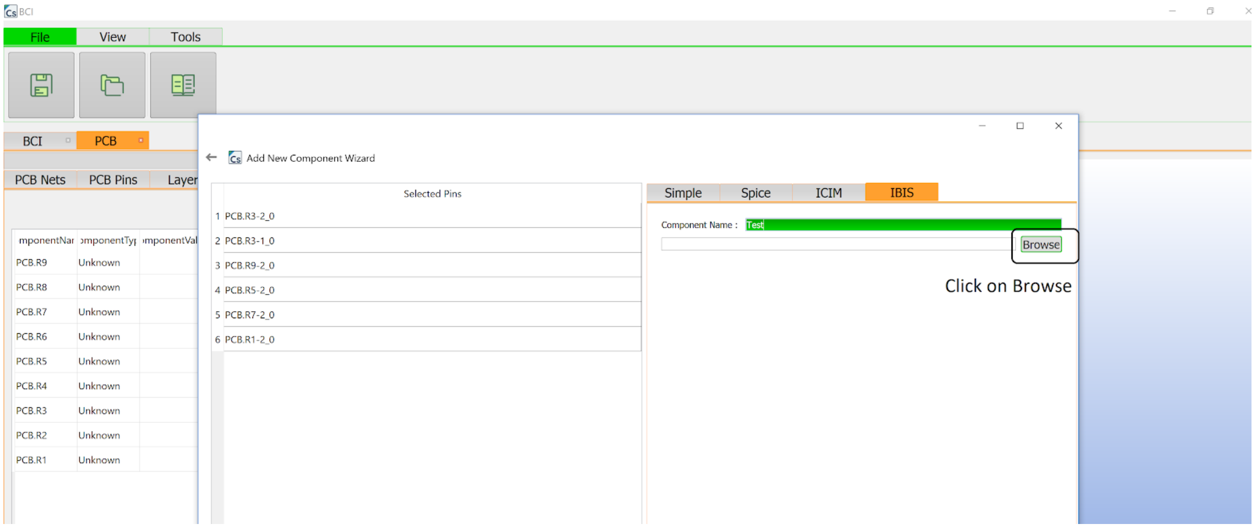


Figure 6-41: Click on Browse (Step 2)

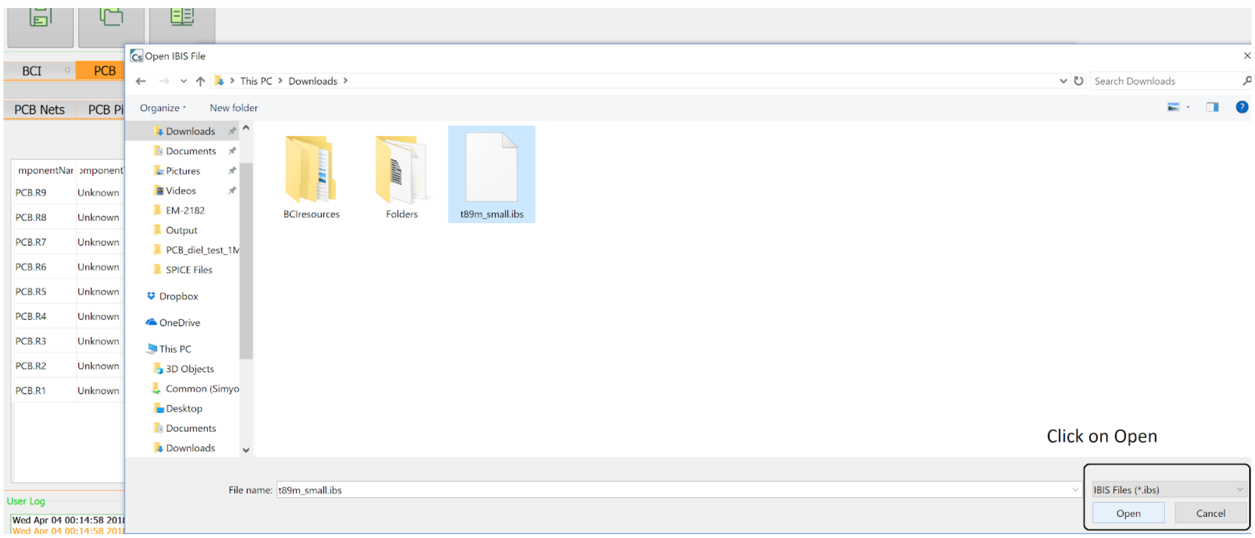


*Figure 6-42: Select the *.ibs file and click Open (Step 2)*

Step 3: Click on Finish

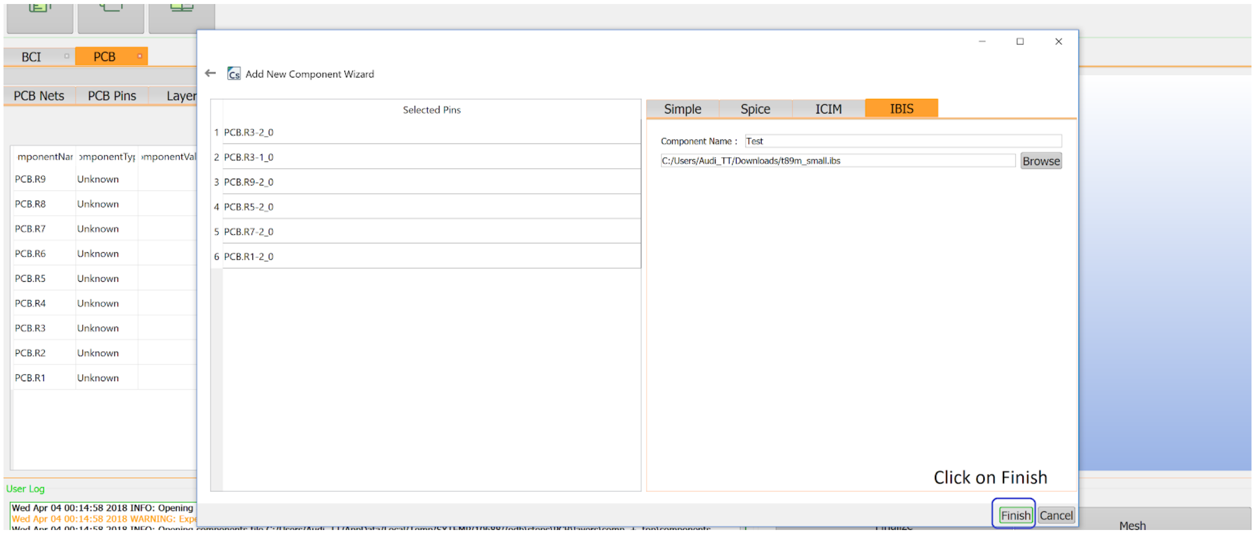


Figure 6-43: Click on Finish (Step 3)

Step 4: It is important to note that, although this does create an IBIS component, that component must still be edited to ensure that the desired configuration is achieved primarily because a single IBIS file generally models families of chipsets instead of a single chip. To edit the component, double click on the created IBIS component. This opens up the edit component window.

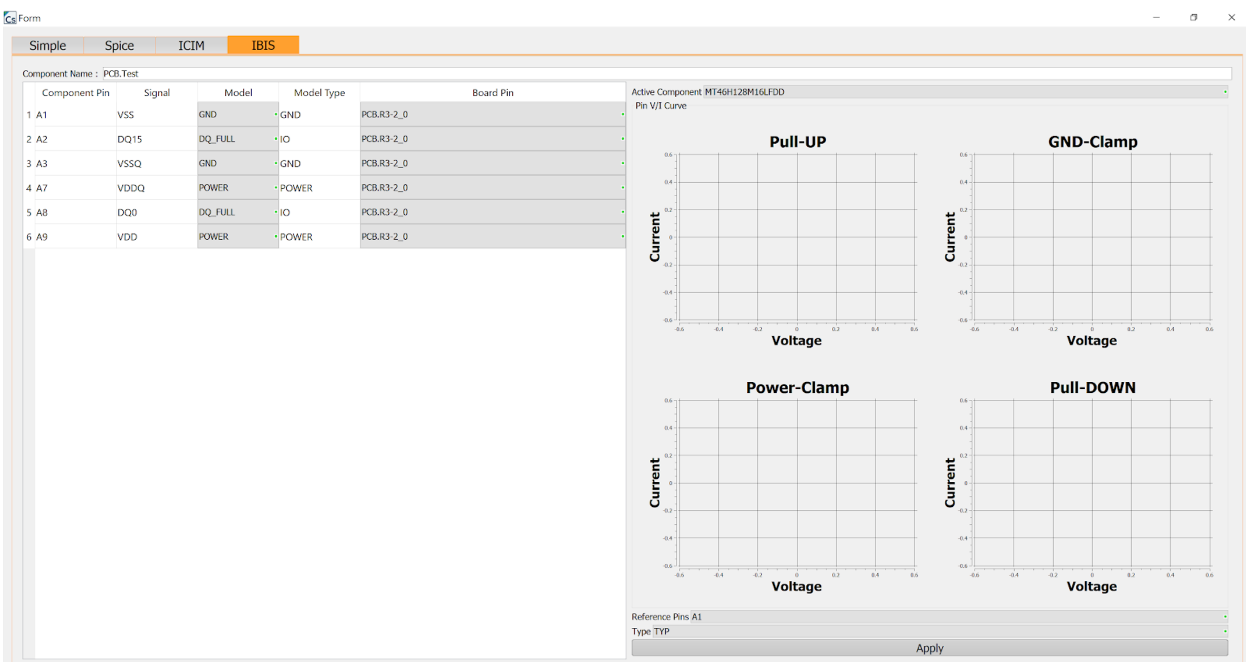


Figure 6-44: The IBIS edit component window (Step 4)

Step 5: The *Active Component* dropdown contains the various components present in the IBIS file. The number of pins required is dependent on the component selected.

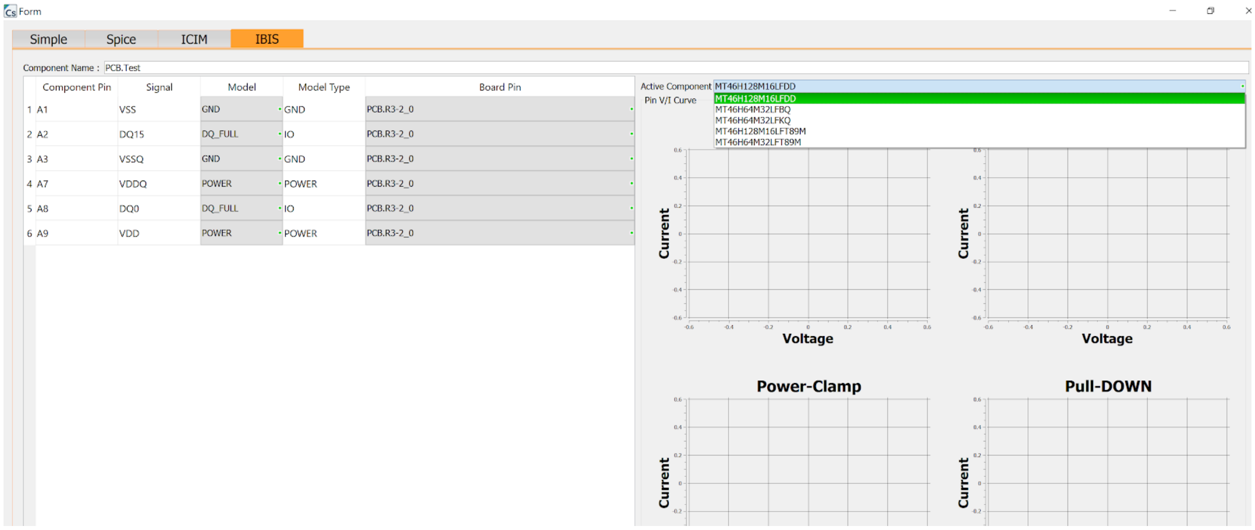


Figure 6-45: The Active Component selection dropdown (Step 5)

Step 6: The model of each pin can be changed from the Model dropdown. While assigning the board pins from the Board Pin dropdown to the Component Pins, it is essential to not assign duplicate pins. Each component pin **must** be mapped to only one,unique board pin.

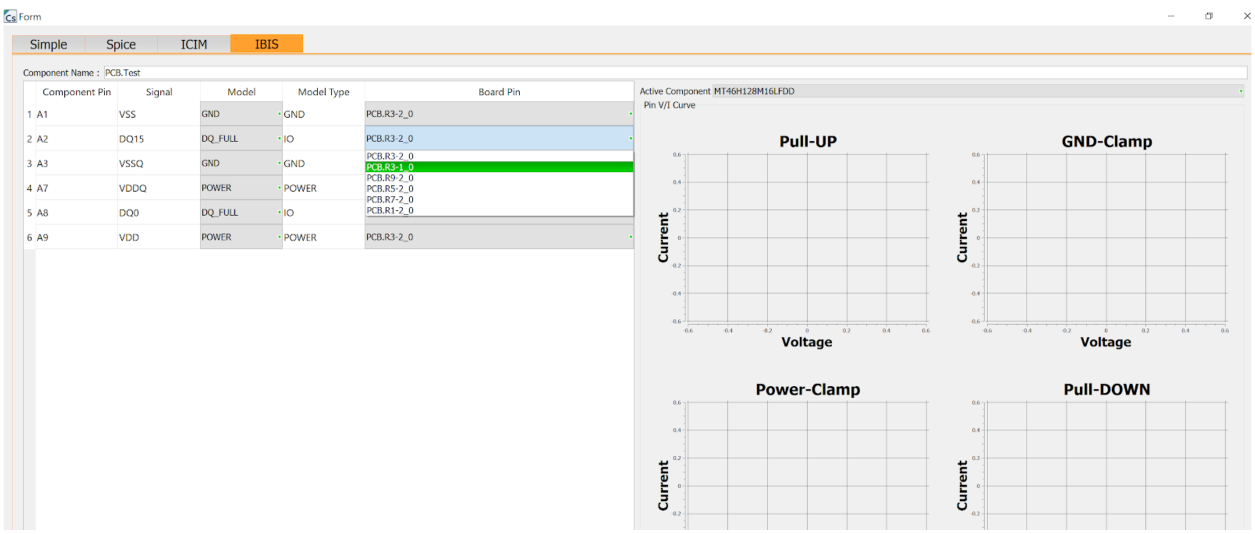


Figure 6-46: Assigning Board Pins (Step 6)

Step 7: The IV curves for pins apart from ground and power can be viewed by selecting the pins from the *Component Pin* column on the table to the left.

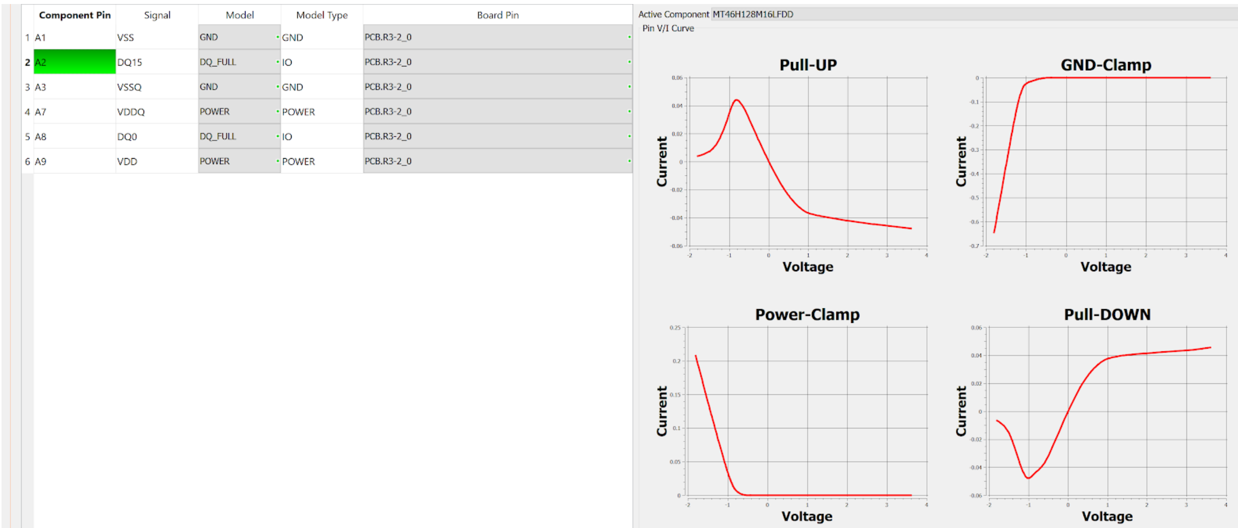


Figure 6-47: Viewing the IV curves for the Pins (Step 7)

Step 8: While computing the IV curves through measurement/simulation during the generation of the IBIS model, a reference pin is paired to every subject pin. Similarly, the IBIS Component edit window allows the user to select the appropriate reference pin from the Reference Pin drop down. The contents of this dropdown might change upon changing the Active Component.

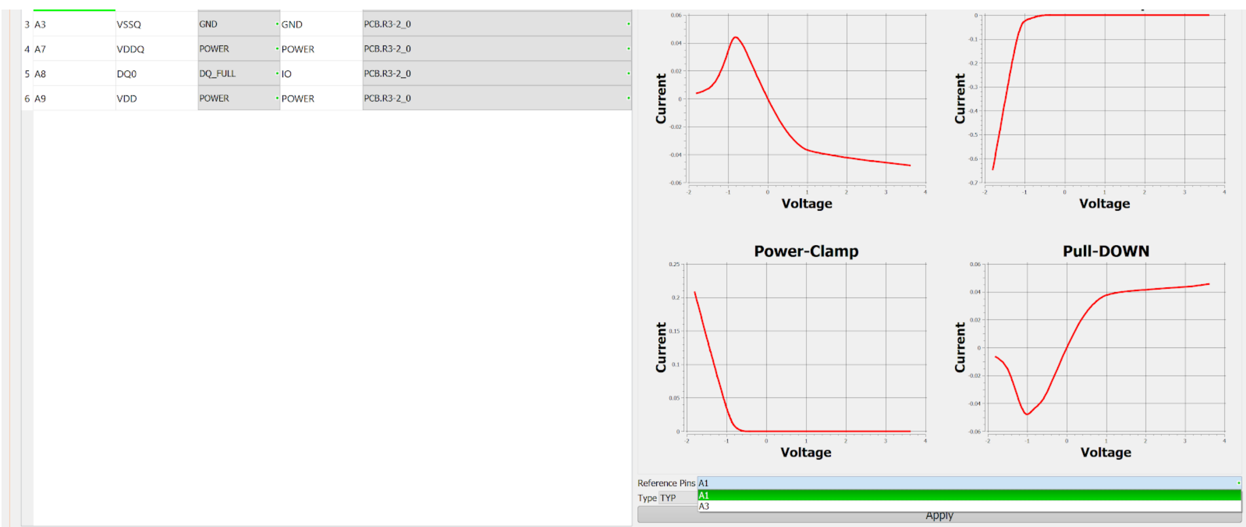


Figure 6-48: Selecting the Reference Pins (Step 8)

Step 9: According to the IBIS specification, during the generation of the IBIS model, the IC is maintained at three states. The first state is the optimum state as directed by the datasheet whereas the other two are the extreme states. These are referred to as *Corners* by the IBIS manual. *Compscope* allows the user to choose a corner through the *Type* dropdown. The Type dropdown has three contents: min, typical and max. The waveforms can change upon changing the Corners.

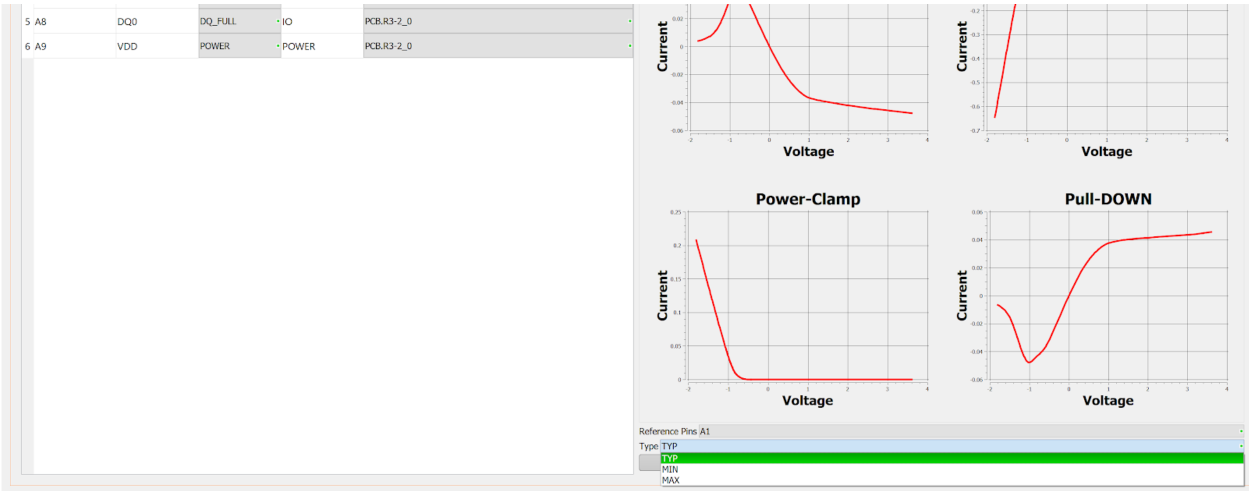


Figure 6-49: Selecting the Corners (Step 9)

Step 10: Once the editing is finished, click on **Apply** and close the window.

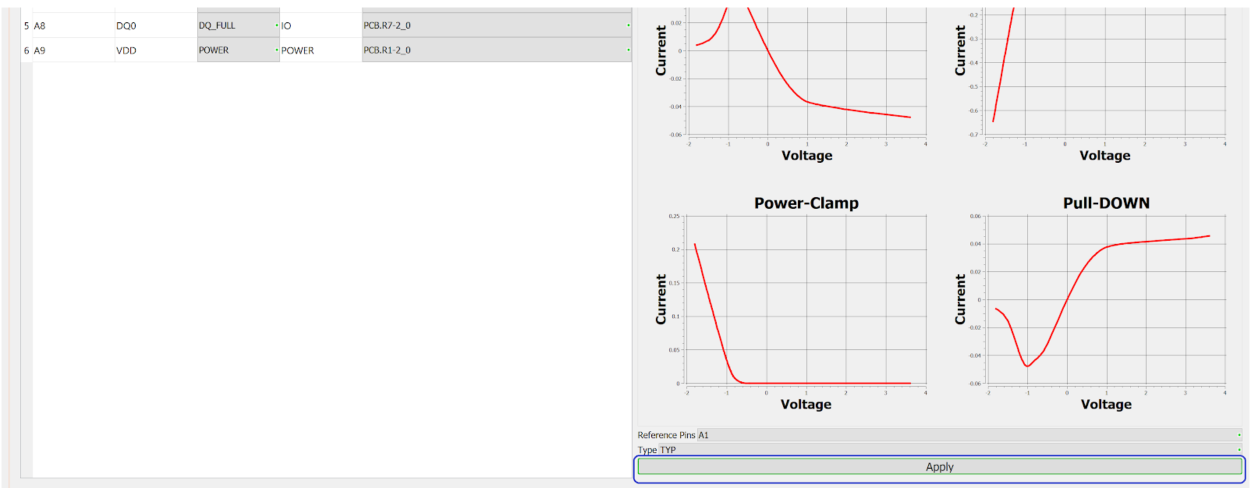


Figure 6-50: Click on Apply (Step 10)

6.10.6.6 Importing BOM for components

Components existing in ODB++ file are impoerted by CompScope as empty components. They can be populated by supplying a BOM file in .csv (Comma Separated Value) format. BOM file format is given below –

Component Name	Description	Value
L1	Inductor	1nH
C2	Capacitor	1mF
R1	Resistor	1KOhm
R2	Resistor	4.7e2

Note that names of component in csv has to match with once which are present on ODB++ file. Press the Load BOM button at top right corner of Component Table a browse dialog to select csv file will appear. Once csv loaded component values, type will get populated as shown in Figure 6-46. In order to probe and enable these components user should manually check the corresponding checkboxes.

Note – Spice sub-circuit, Nport data and IC-model import as BOM is not currently supported. Only RLC components can be populated as .csv import.

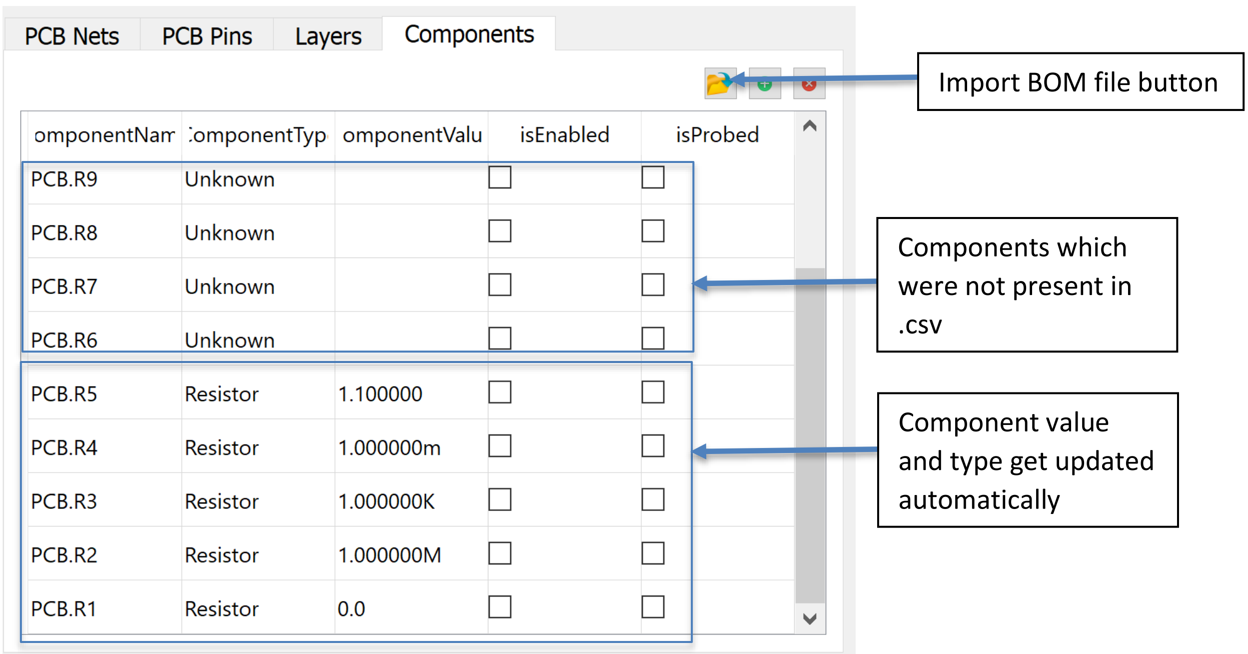


Figure 6-46: Components import from BOM

6.10.6.7 Deleting a component

To delete component select corresponding row in Component Table and click on Delete button at top right corner of the Table.

7 Chapter 7: IC models

This chapter discusses the different forms of IC models that are supported in CompScope. Currently, only the ICIM-CI is discussed, since it is used in BCI.

7.1 Why do we need a special type of model for ICs?

Two different methodologies for supporting a component model has been described in 6.10.6 Components:

(a) Simple Component Model: As shown in 6.10.6.2 "Simple Component" can be used to model a Resistor, Capacitor, Inductor etc

(b) SPICE/Touchstone Component Model: As shown in 6.10.6.3 "SPICE and Nport" block may be used to import a component.

A complete black-box representation of the IC is difficult to generate:

(a) Why not a complete S-parameter block: The IC contains non-linear elements, so it's transfer functions cannot be represented by a Linear, Time Invariant (LTI) black-box.

(b) Why not a complete SPICE sub-ckt: This is possible in theory. However, the IC manufacturer will not be willing to share the detailed information of the internals of the IC. Also, most ICs contain more than million transistors along with parasitic elements which will be extremely time consuming to process in a BCI simulation.

Therefore, CompScope follows the IEC standards defined for conducted/radiated immunity/emission models.

7.2 For BCI: ICIM-CI

Please refer to 13.1 BCI: IEC-62433-4 Standard for a description of the IEC model before reading below.

7.2.1 Creating an ICIM component

An ICIM component can be created in the same way as a "Simple" or "SPICE" component.

Step 1: Click the "Add New Component" button as shown in Figure 7-1.

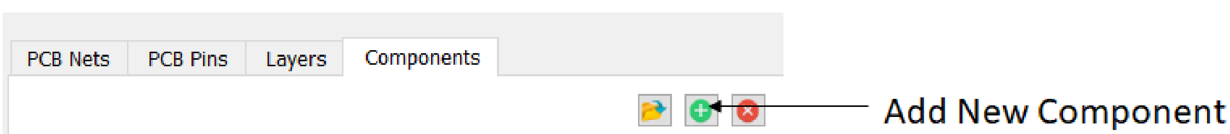


Figure 7-1: Add new component button

Step 2: Select the pins for the IC as shown in Figure 7-2.

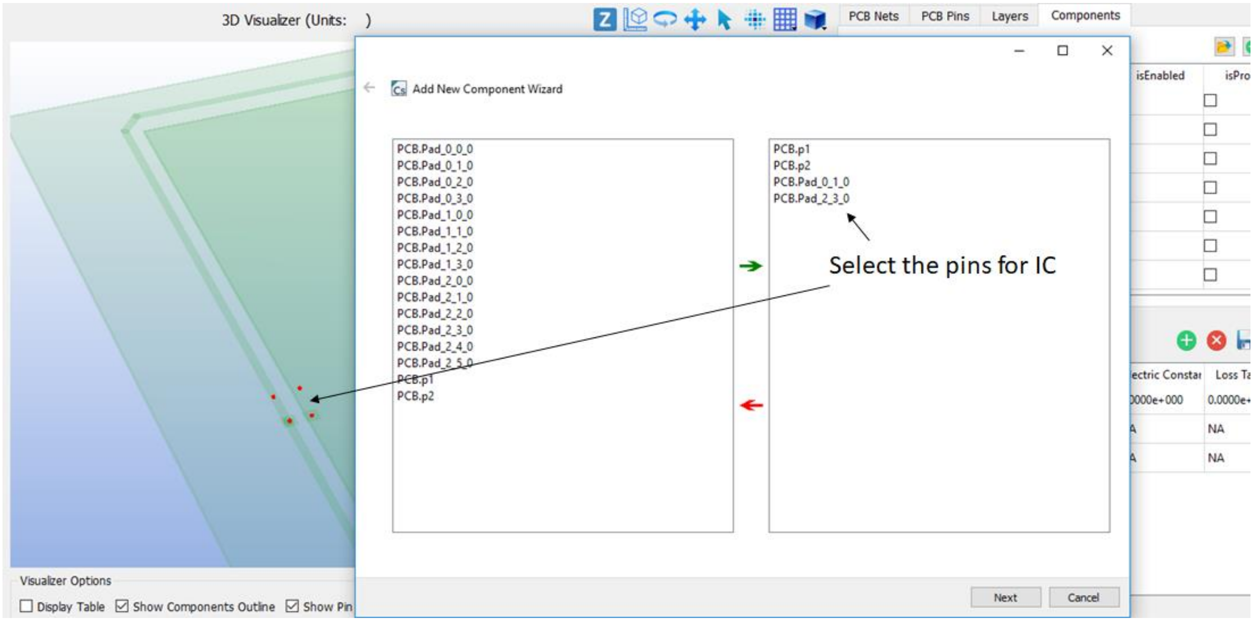


Figure 7-2: Selecting the pins for the IC

Step 3: Click “Next”

Step 4: Select “ICIM”, give a name and finish

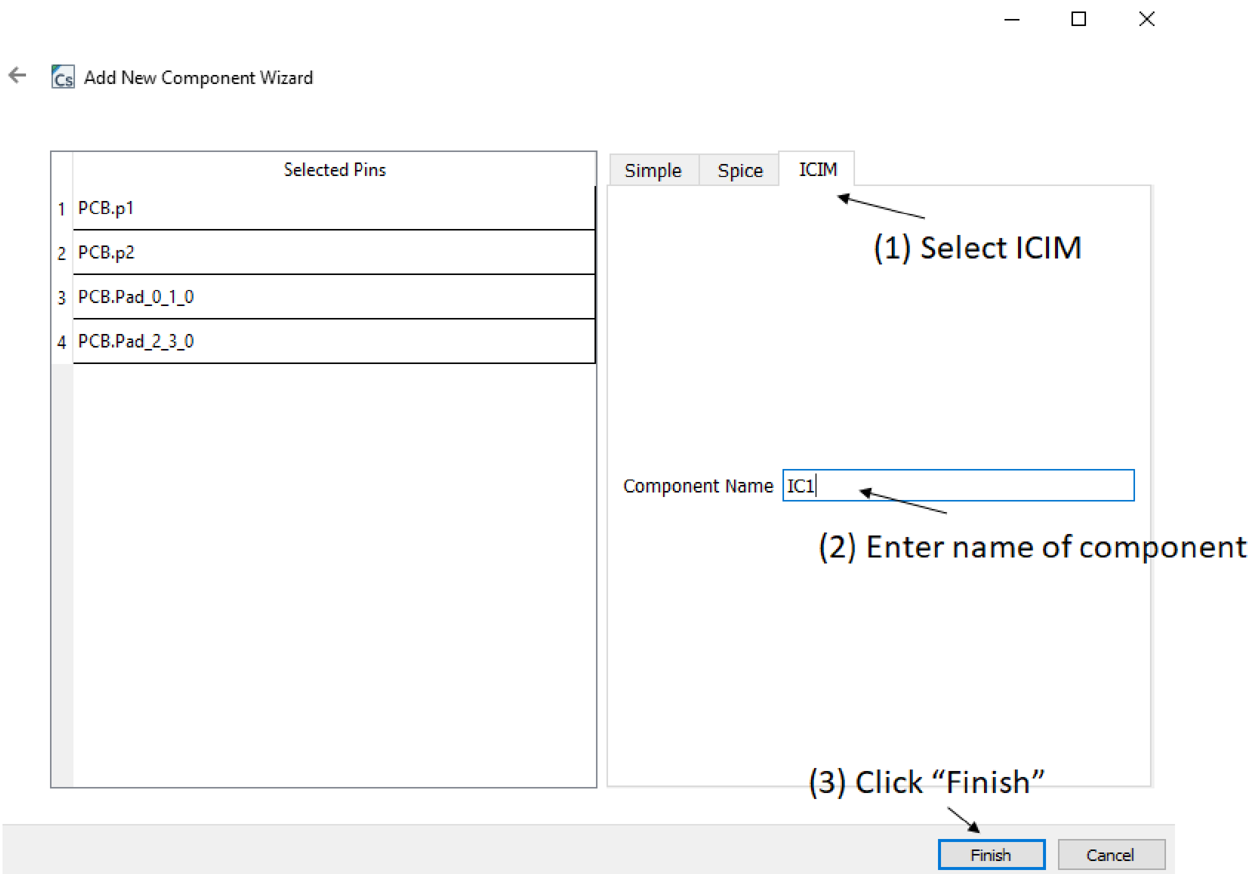


Figure 7-3: Create an ICIM component

The component is now created and can be seen under the component table as shown in: Figure 7-4.

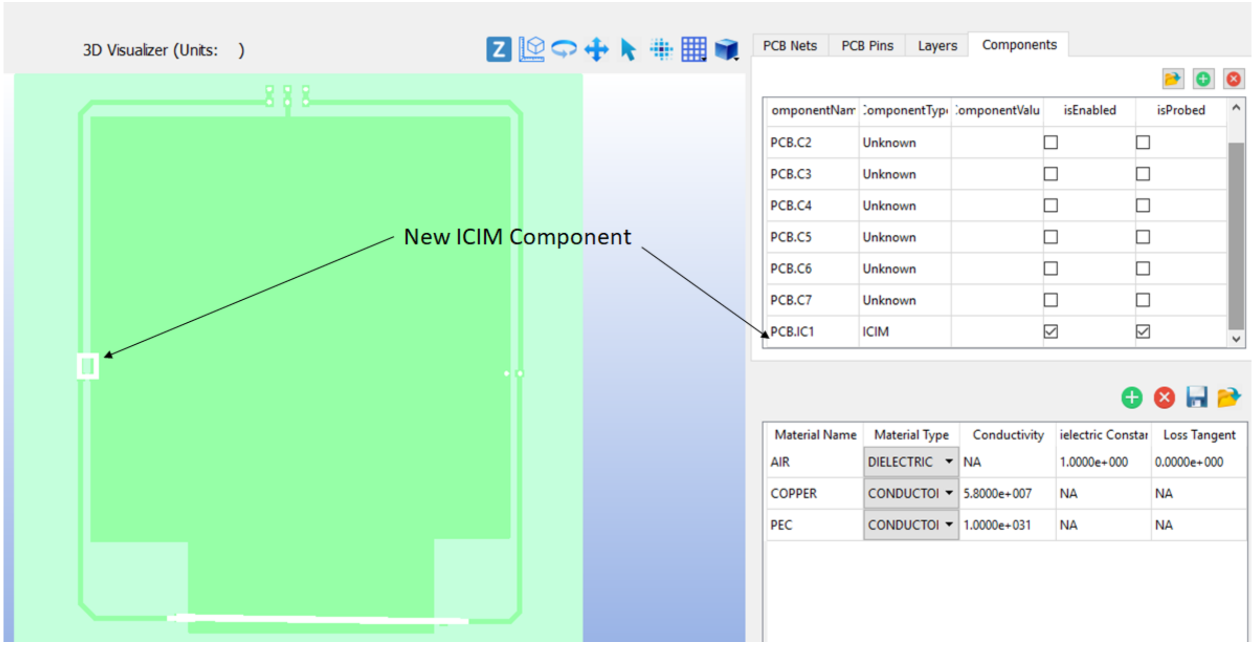


Figure 7-4: New ICIM component

7.2.2 Editing an ICIM Component

On double-clicking the component, the editing form opens:

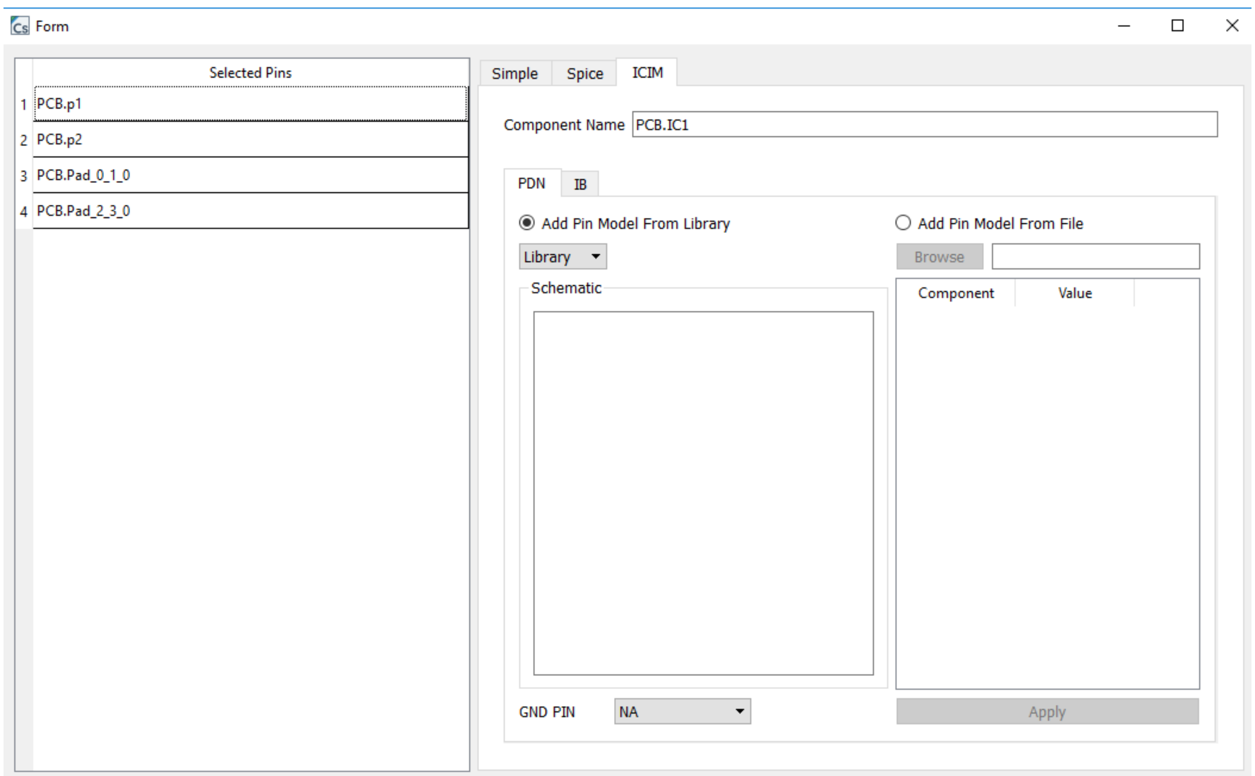


Figure 7-5: Editing window for ICIM component

As discussed in 13.1 BCI: IEC-62433-4 Standard, the main components of the ICIM model is the "Passive Distribution Network" (PDN) and "Immunity Behavior" (IB).

7.2.2.1 PDN

In this section, importing data for the PDN section of ICIM is described. In general all pins of the IC can be assigned an independent PDN. Although, IEC-62433-4 supports single-pin, differential-pin and multi-pin models, current version of CompScope only supports single-pin models. In order to populate data for any given pin of the IC:

Step 1: Select the pin from the pins window as shown below

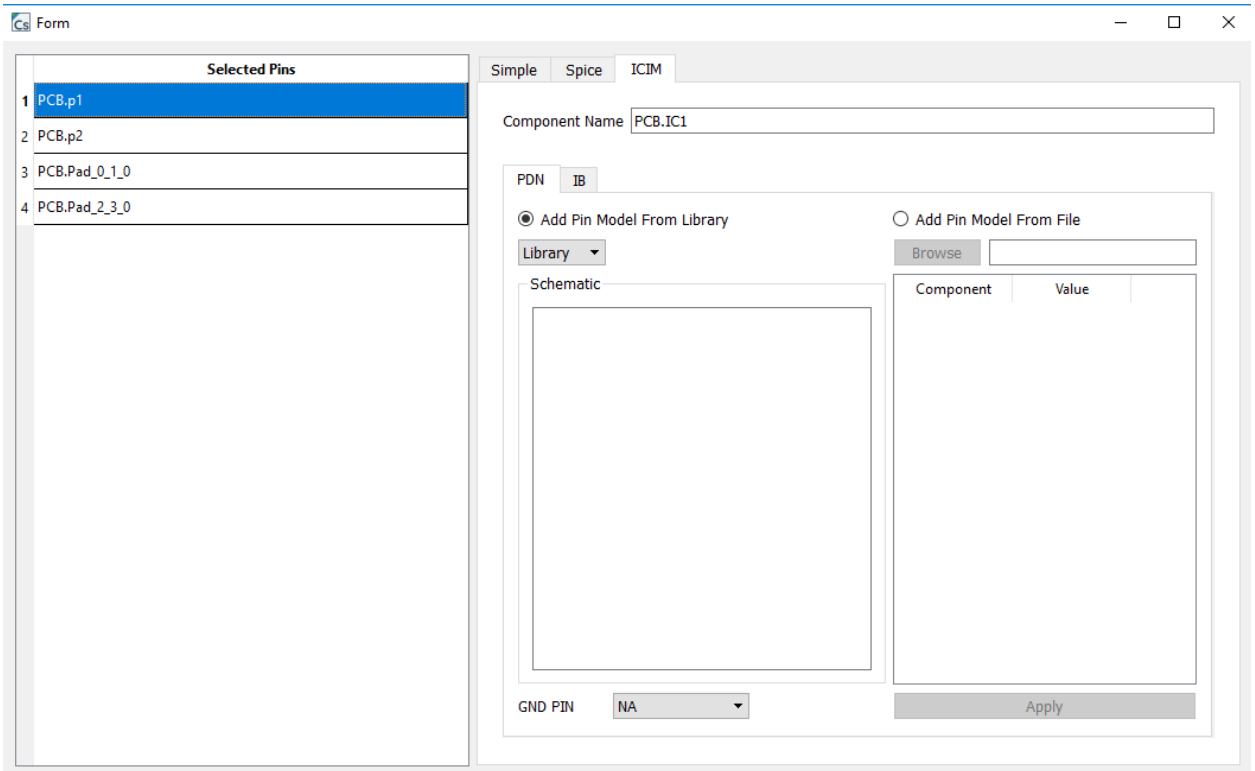


Figure 7-6: Select a pin for ICIM PDN assignment

Step 2: Assigning a PDN to the selected pin can be done in 2 different ways:

(a) Loading from a *.s1p or *.sp/*.cir file using the "Add Pin Model From File" radio-button as shown in Figure 7-7.

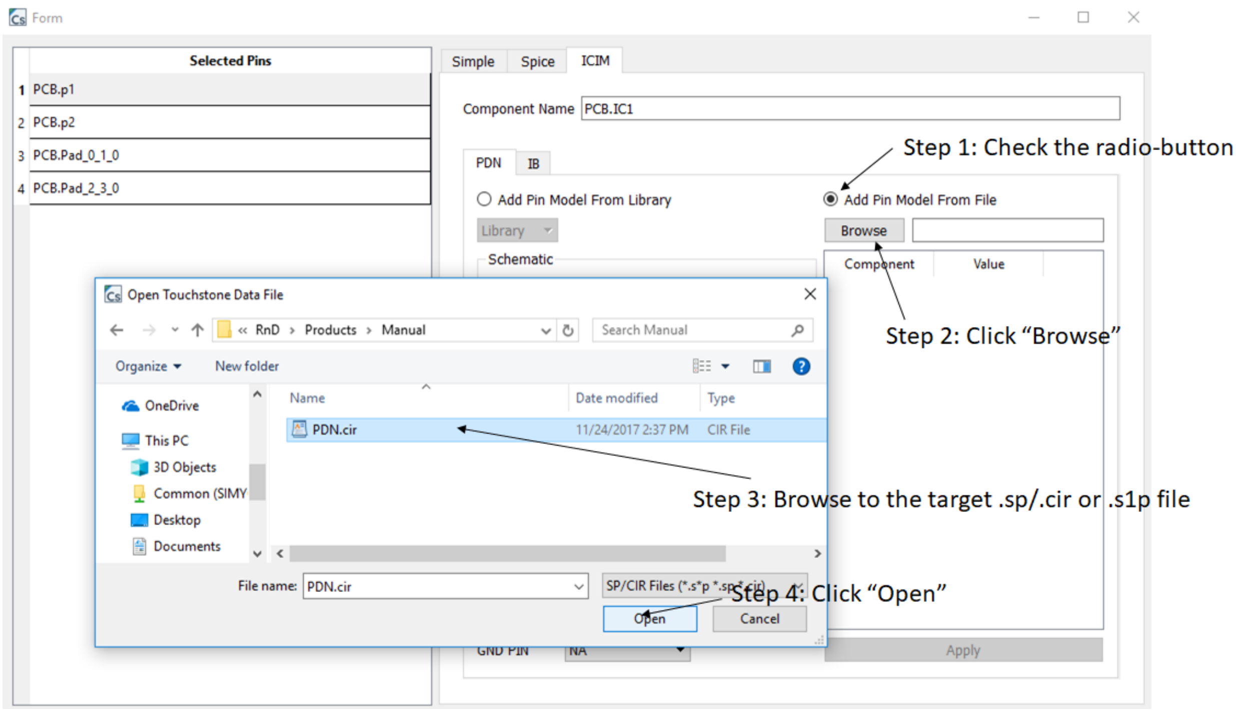


Figure 7-7: Importing a file for a PDN of ICIM

Once imported the contents will show up in the "Schematic Box":

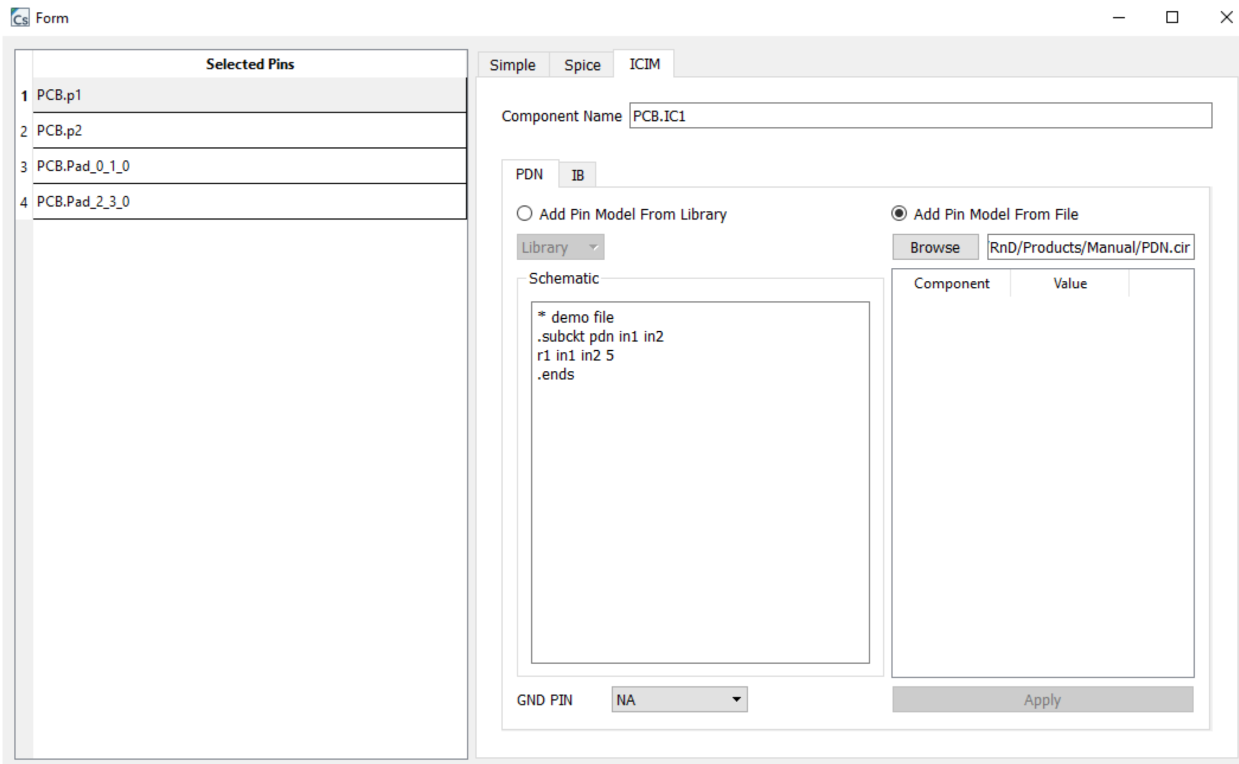


Figure 7-8: Loaded ICIM PDN in the Schematic box

(b) Loading the PDN from library:

Currently, there is one library model available – DigitialIO. Many of the digital IC pins end up in this protective circuit, therefore this model is enabled in the library. In order to assign a pin PDN with this library model, select the “Add Pin Model from Library” radio-box and choose the DigitalIO drop-down.

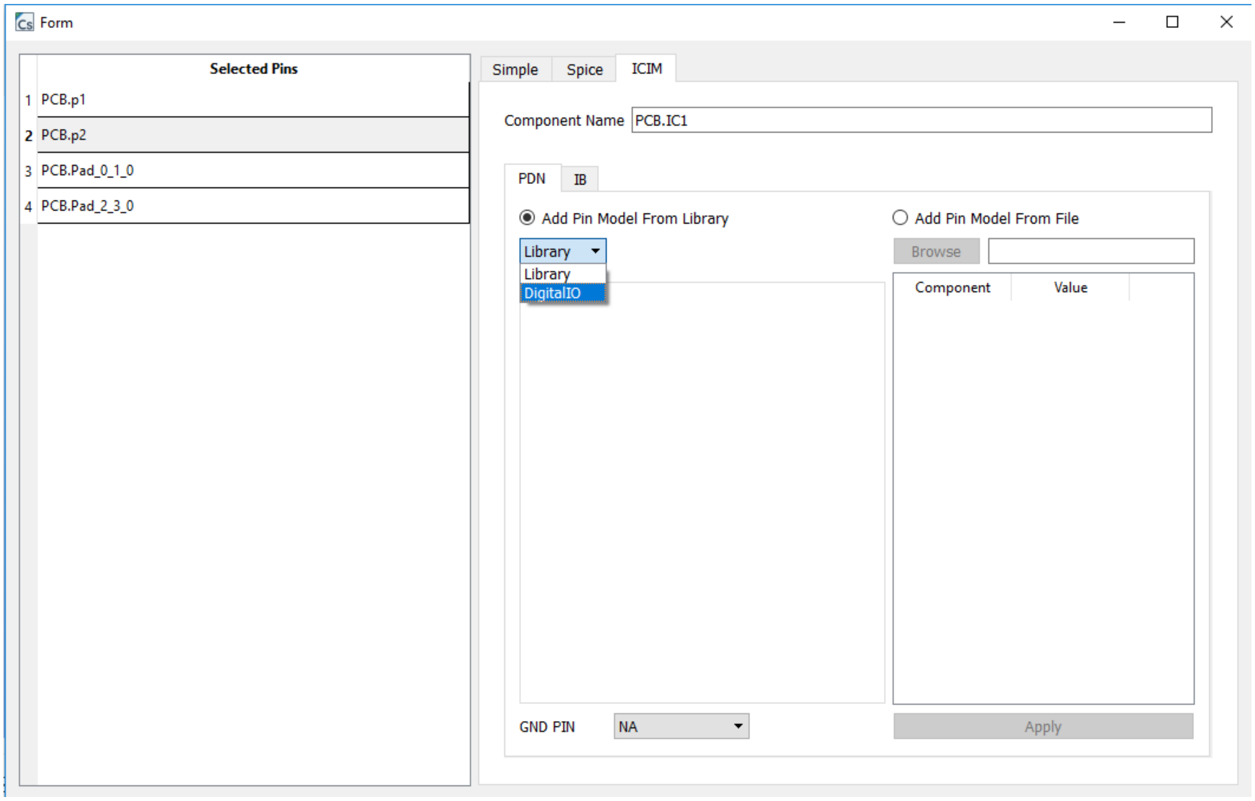


Figure 7-9: Selecting the DigitialIO library model for the PDN

Complete the process, by selecting the corresponding ground pin for the selected pin, editing desired values for the schematic and click apply as shown in Figure 7-10.

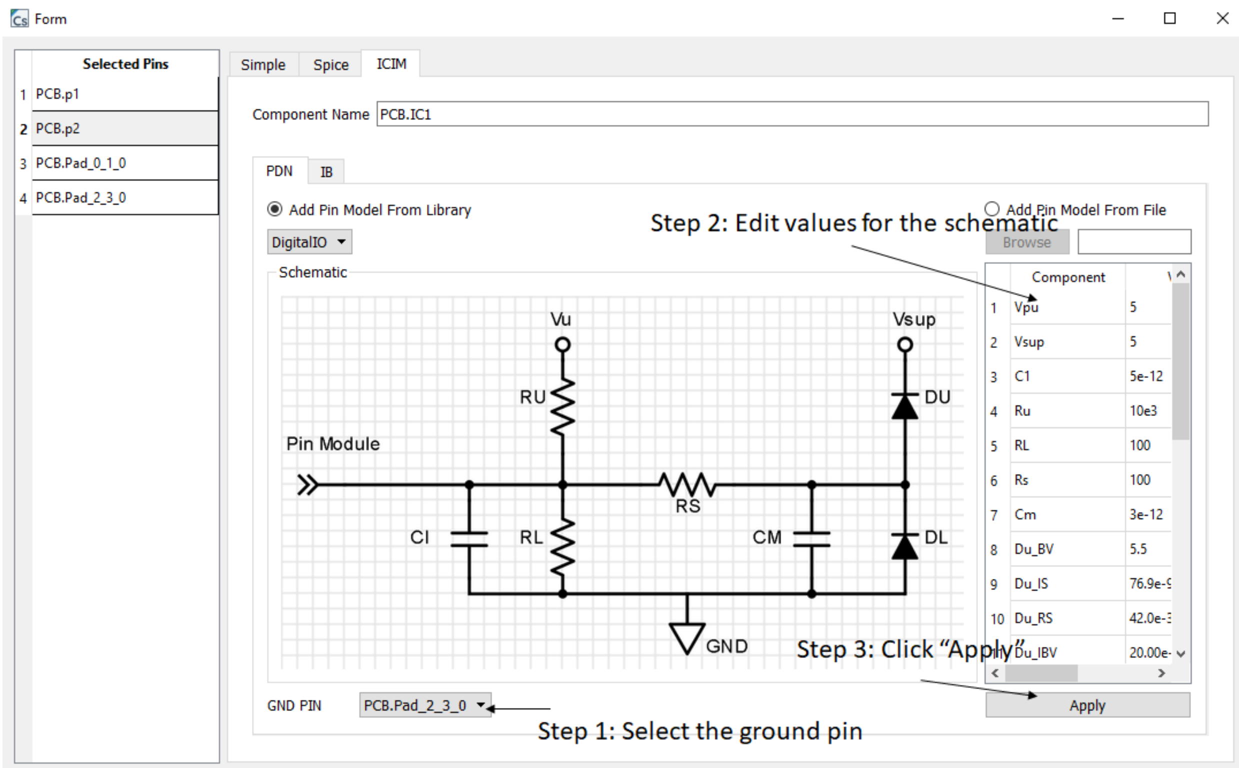


Figure 7-10: Assigning the digitalIO model to pin

This will assign the schematic netlist to the selected pin. The corresponding text will be visible in the "Schematic Box" as shown in Figure 7-11.

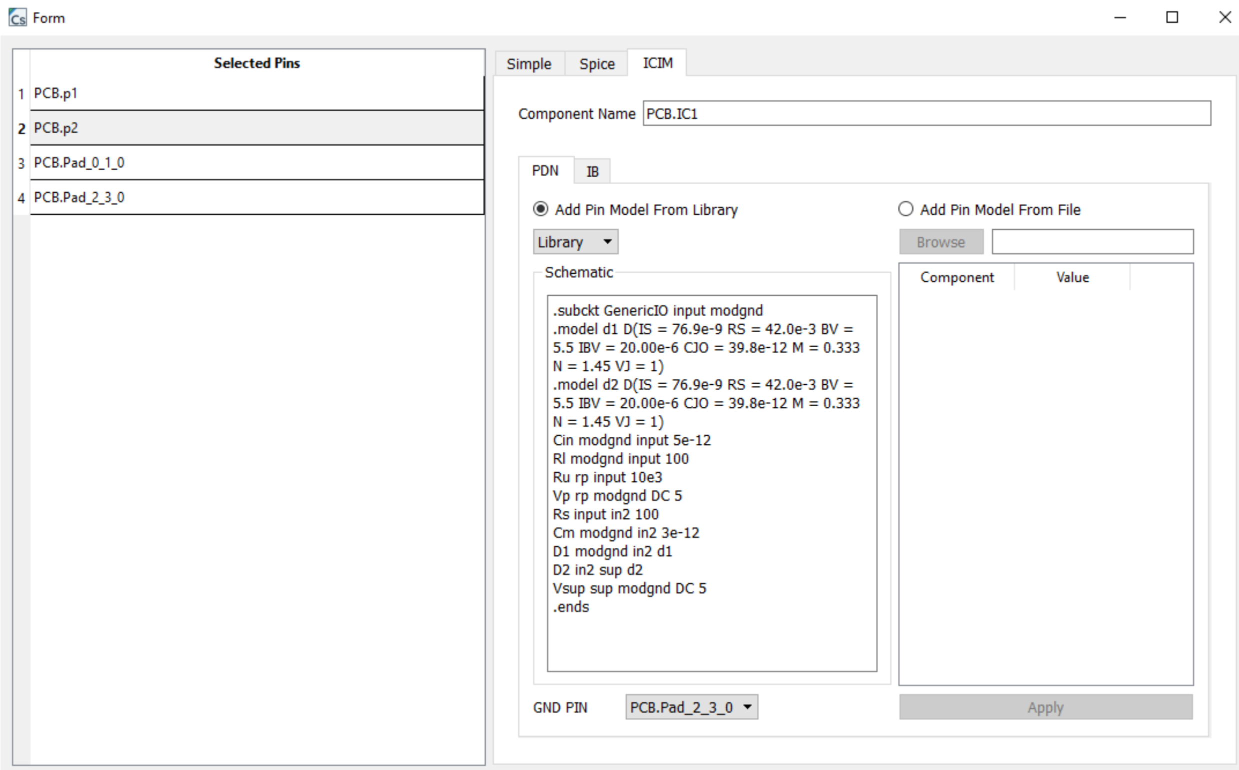


Figure 7-11: DigitalIO PDN subcircuit

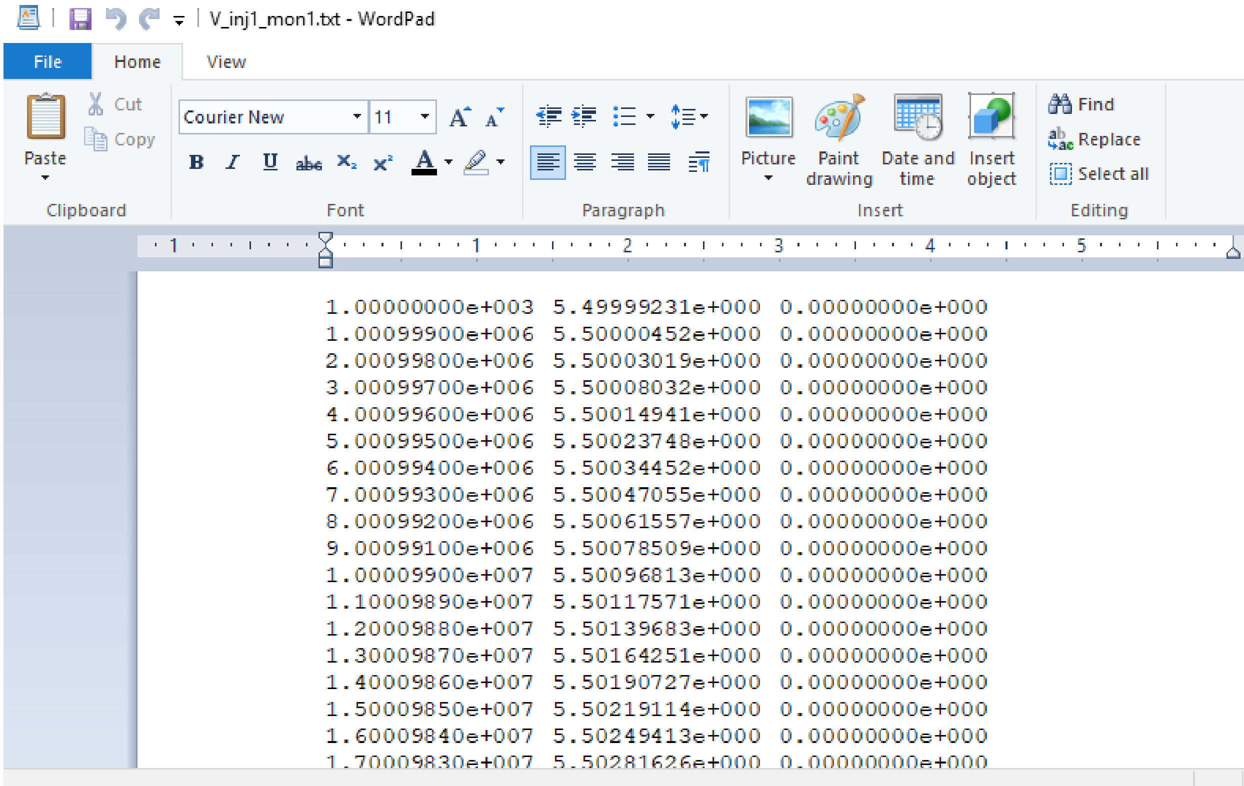
7.2.2.2 IB

The "Immunity Behavior" is effectively the limit-line for noise on the selected pin. It can be defined in the form of a voltage, current or power profile. Therefore an IB is nothing but a frequency vs value data. It must be noted that the IB does not affect CompScope simulation but only helps in generating the pass-fail report after the noise computation have been performed for the experiment.

An IB can be formed in 2 different ways:

(a) Loading from a file:

A typical IB file is shown below:



Frequency	Value_Real	Value_Imaginary
1.00000000e+003	5.49999231e+000	0.00000000e+000
1.00099900e+006	5.50000452e+000	0.00000000e+000
2.00099800e+006	5.50003019e+000	0.00000000e+000
3.00099700e+006	5.50008032e+000	0.00000000e+000
4.00099600e+006	5.50014941e+000	0.00000000e+000
5.00099500e+006	5.50023748e+000	0.00000000e+000
6.00099400e+006	5.50034452e+000	0.00000000e+000
7.00099300e+006	5.50047055e+000	0.00000000e+000
8.00099200e+006	5.50061557e+000	0.00000000e+000
9.00099100e+006	5.50078509e+000	0.00000000e+000
1.00009900e+007	5.50096813e+000	0.00000000e+000
1.10009890e+007	5.50117571e+000	0.00000000e+000
1.20009880e+007	5.50139683e+000	0.00000000e+000
1.30009870e+007	5.50164251e+000	0.00000000e+000
1.40009860e+007	5.50190727e+000	0.00000000e+000
1.50009850e+007	5.50219114e+000	0.00000000e+000
1.60009840e+007	5.50249413e+000	0.00000000e+000
1.70009830e+007	5.50281626e+000	0.00000000e+000

Figure 7-12: Typical IB file

Although, in most cases, the magnitude of the profile is given, the file format has a provision for the imaginary component. In order to assign this file as IB, the user needs to:

(i) Select the aggressor pin

(ii) Select the victim pin

(iii) Select the profile type

(iv) Click “Add” to browse and IB file

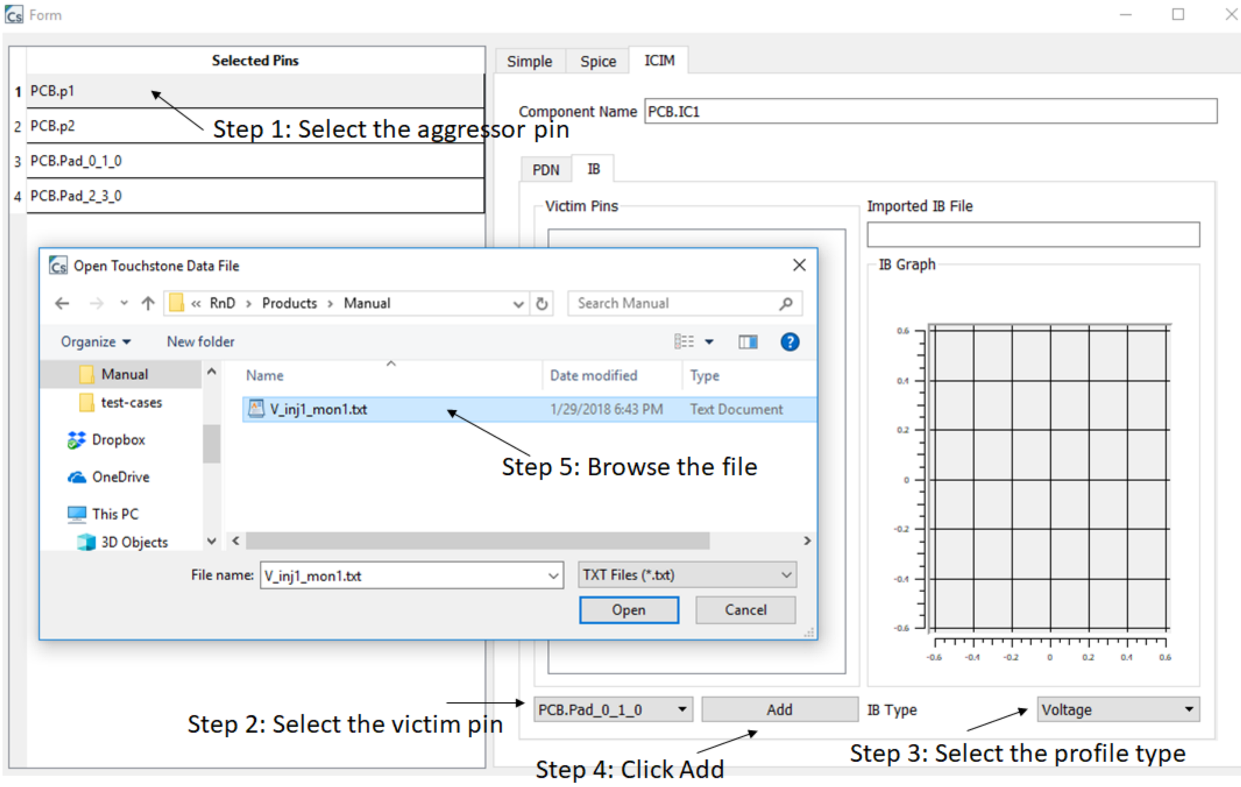
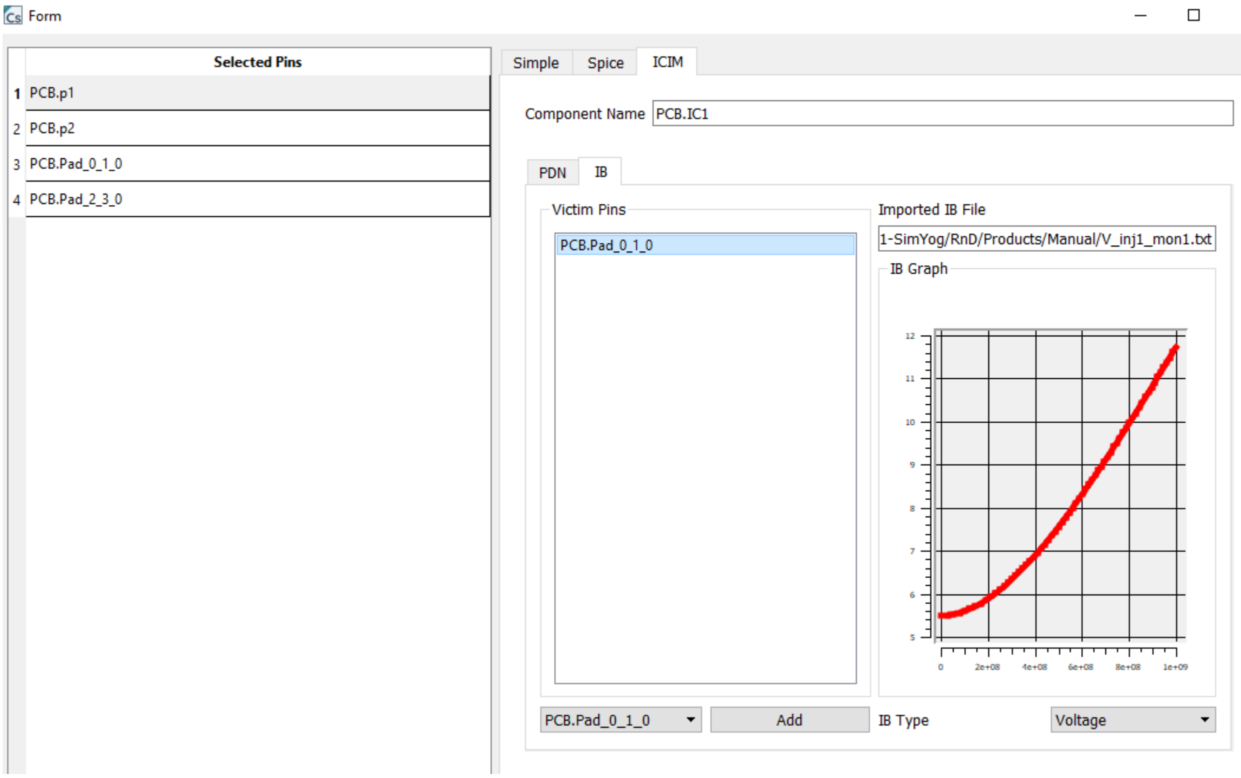


Figure 7-13: Adding an IB profile using imported file

Once the file has been imported, the profile should be visible on the graph when the victim pin is highlighted:



It should be noted, that the user can add multiple IB for a given aggressor/selected pin for different victim pins by repeating the process described above:

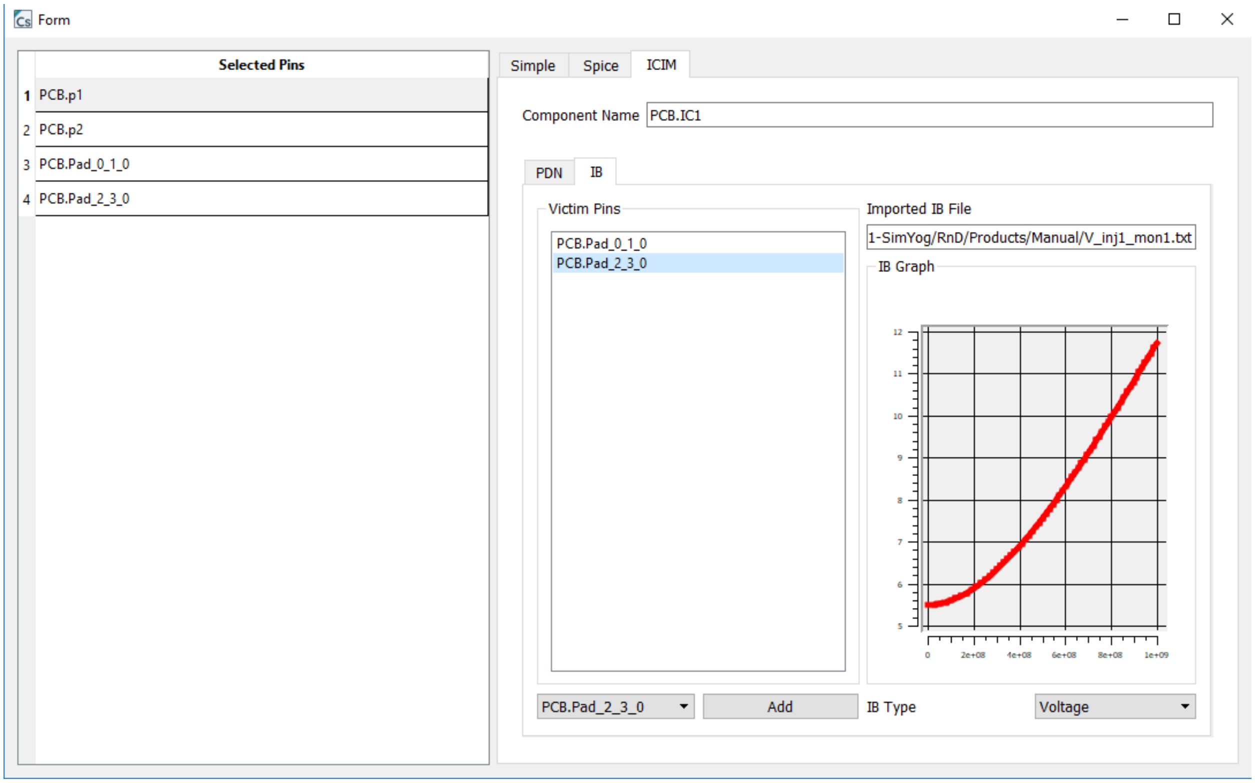


Figure 7-14: Multiple victim pins is permissible for a given aggressor/selected pin

(b) Automatic IB assignment on PDN import:

When importing the PDN from library, the tool automatically computes the IB profile or the pass-fail limit for the given PDN. In this case, the victim pin is assigned as the same as the aggressor pin. This can be overwritten by importing user's own file.

If an experiment has one or more IBs defined, the user is able to "Generate Report" (see **Error! eference source not found. Error! Reference source not found.**) after the "Analyze" process is complete. Also, on opening the IB window, the limit-line is shown along with the computed noise profile as shown below:

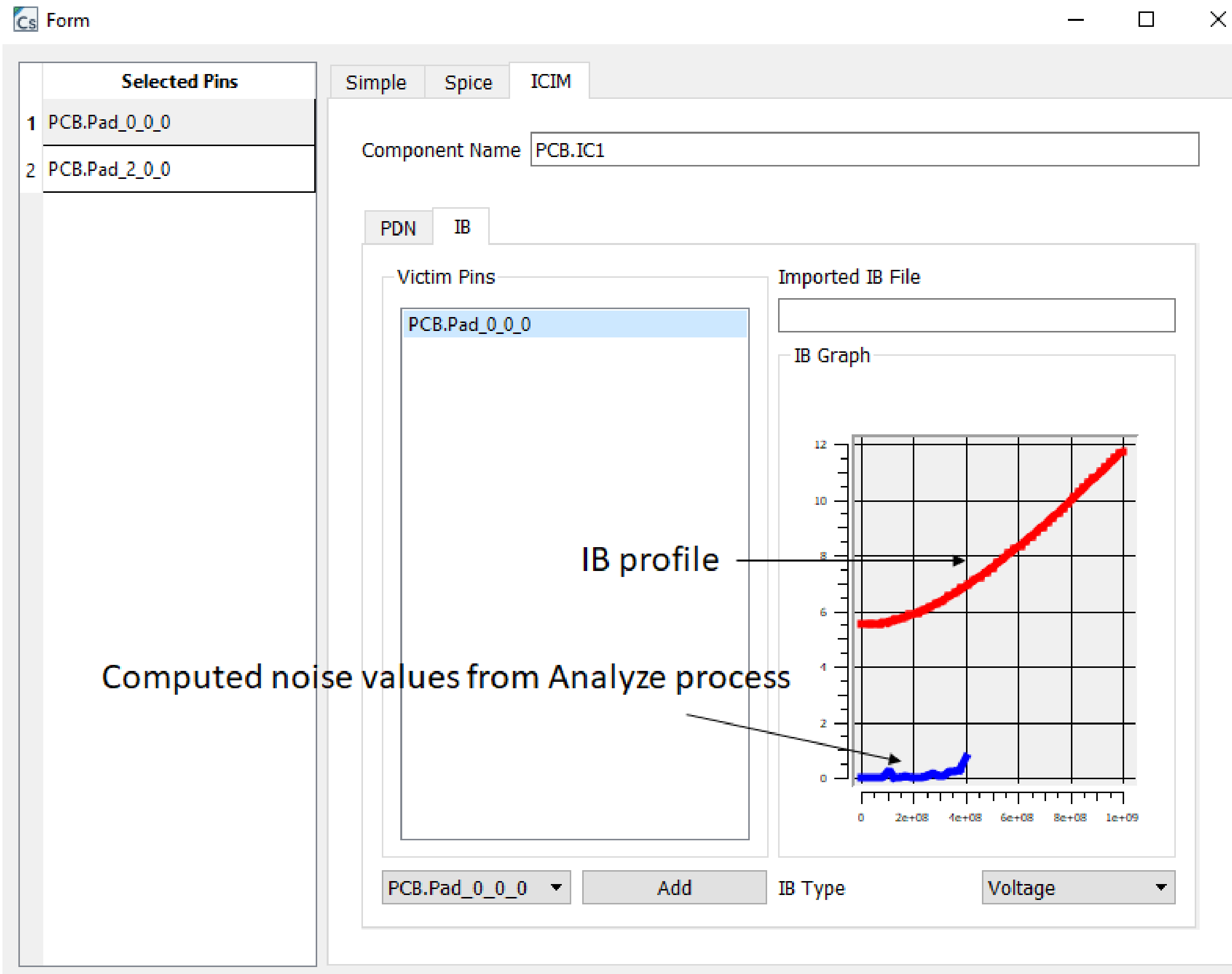


Figure 7-15: Computed noise value and limit-line IB

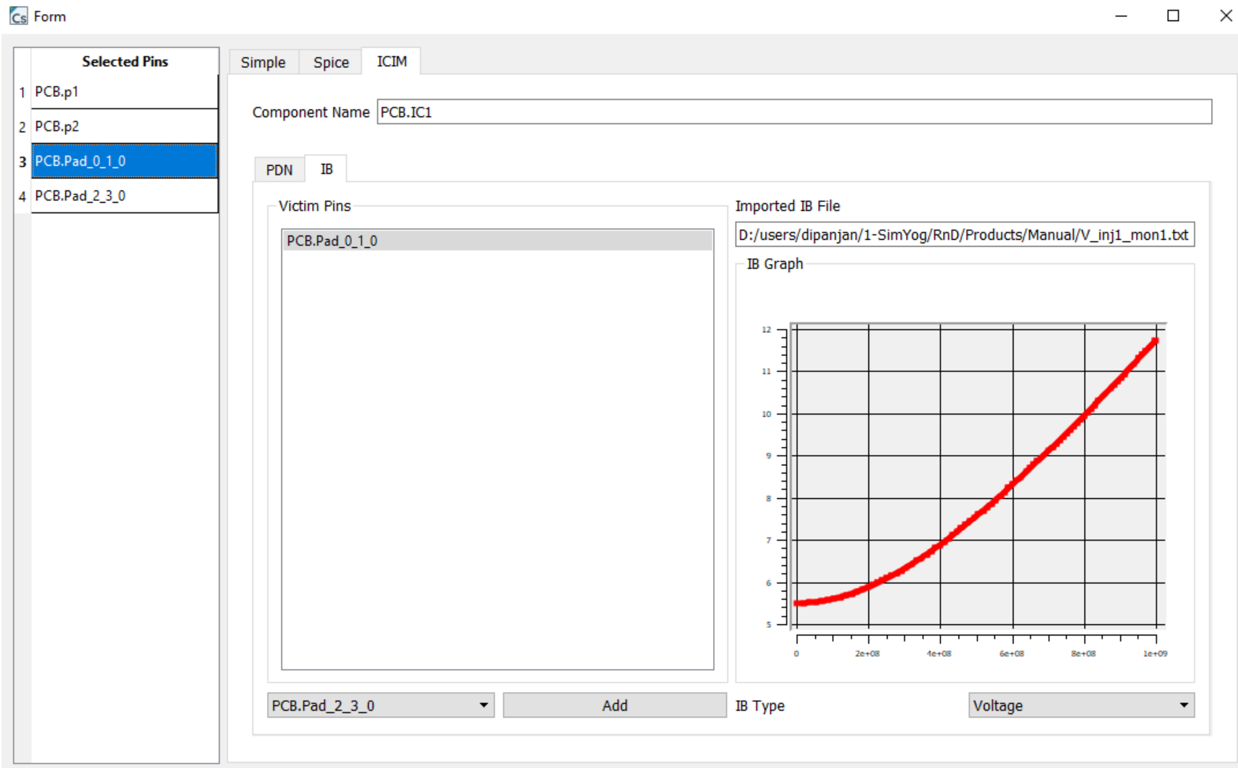


Figure 7-16: Automatic IB formation on loading PDN from library

7.3 For RE: ICEM-CE

CompScope RE supports importing Components in the form of “Integrated Circuit Emission Model – Conducted Emissions (ICEM-CE)” which is an IEC standard (IEC-62433-2). This model describes the Passive Distribution Network (PDN) and the Internal Activity (IA) of the chip.

7.3.1 Creating an ICEM Component

(a) Step 1: On the PCB widget, click “Add New Component”

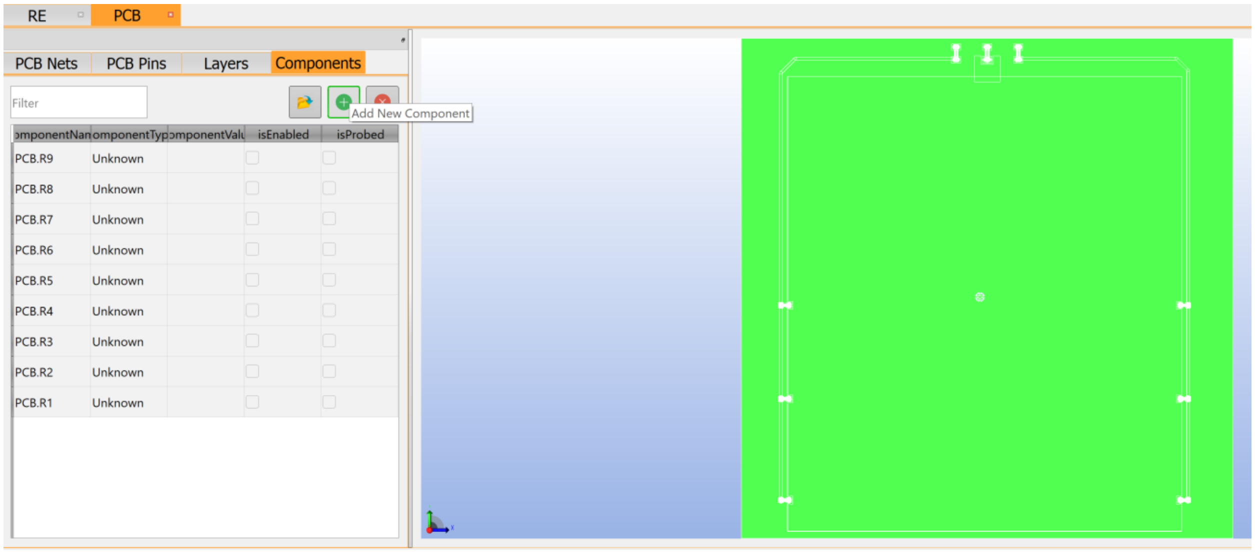


Figure 7-17: Add a component

(b) Select the pins of the IC, then click "Next"

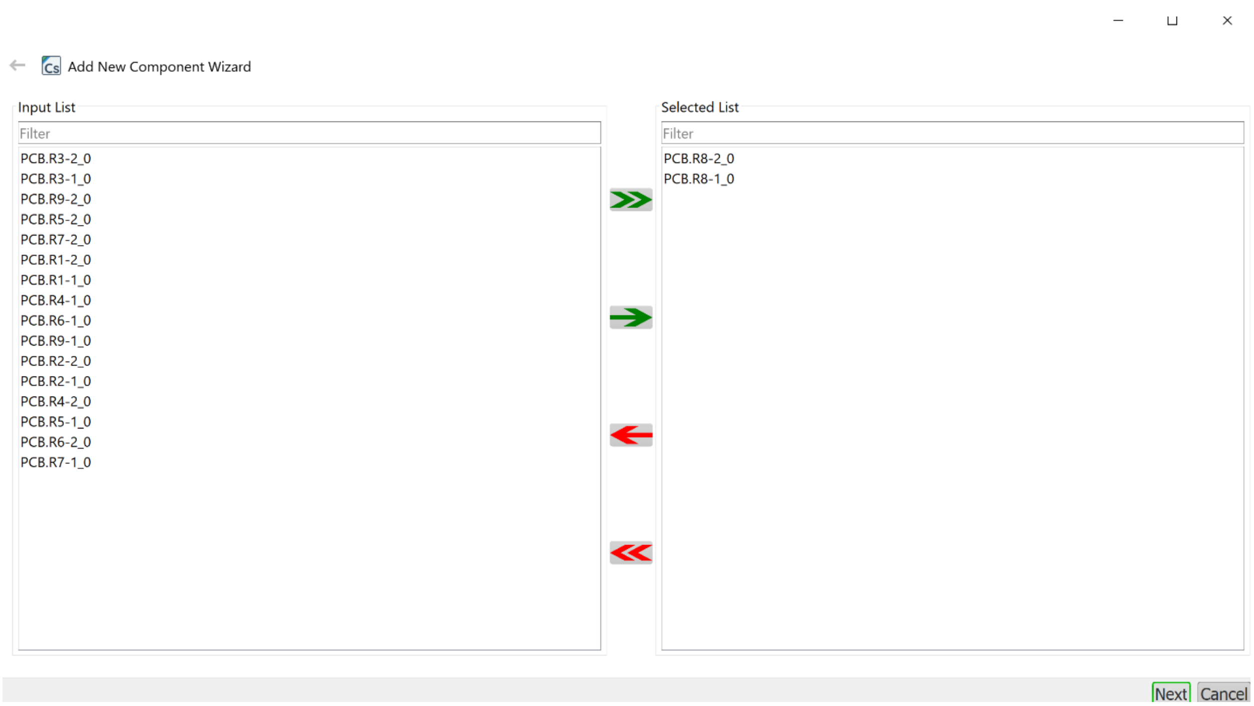


Figure 7-18: Select the pins of the component

(c) Select "ICEM", name the component and click "Finish"

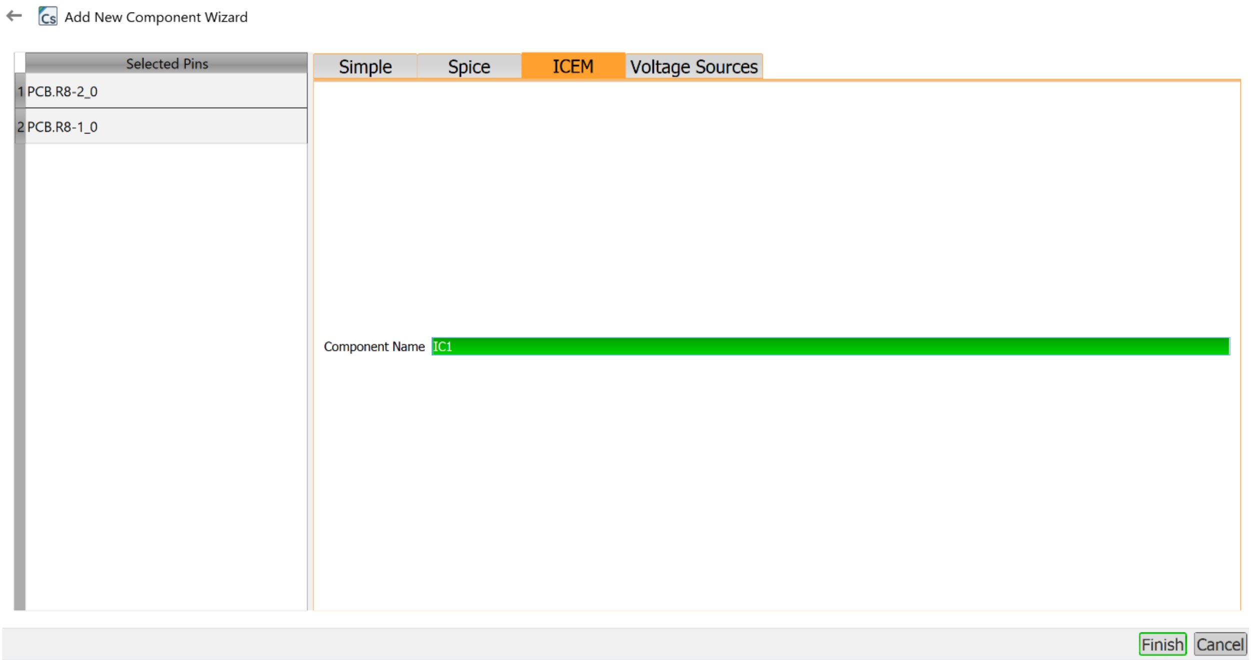


Figure 7-19: Select ICEM, name the component and click Finish

(d) The component will show up as an ICEM component in the component list

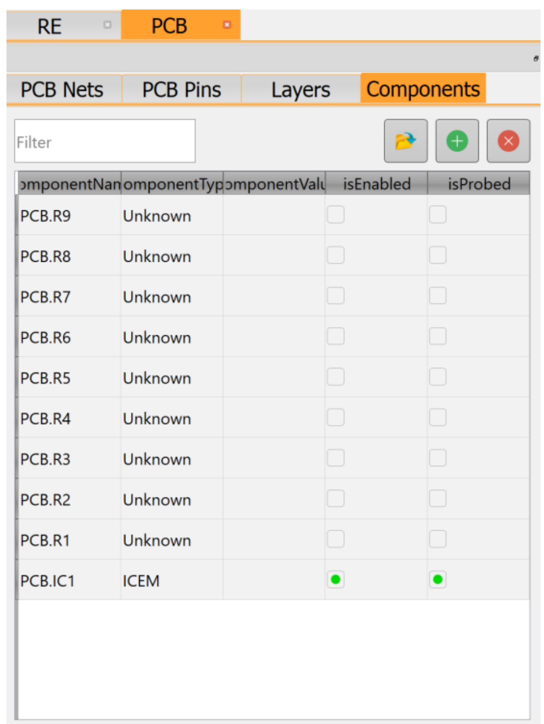


Figure 7-20: Newly added ICEM component in component list

7.3.2 Editing an ICEM Component

The newly created ICEM component is empty when created and needs to be edited to populate the PDN and IA characteristics. To edit the ICEM component double click the component name in the component list to open the editing widget.

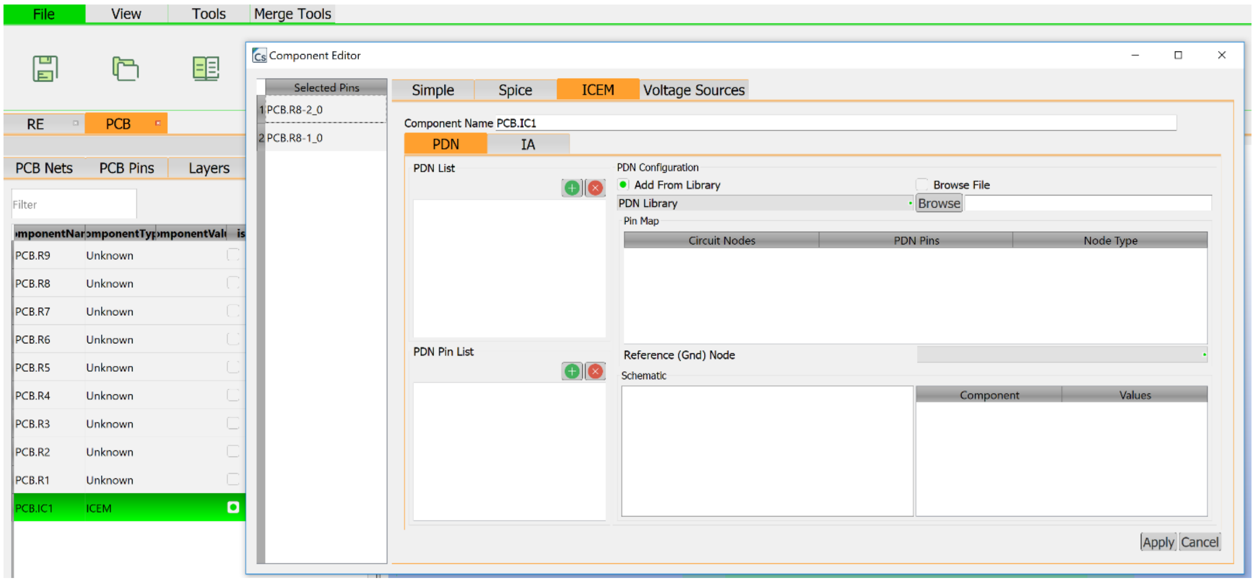


Figure 7-21: ICEM editing widget

7.3.2.1 PDN

(a) A PDN can be added by clicking the "+" button. Multiple such PDNs are supported per component (per IC)

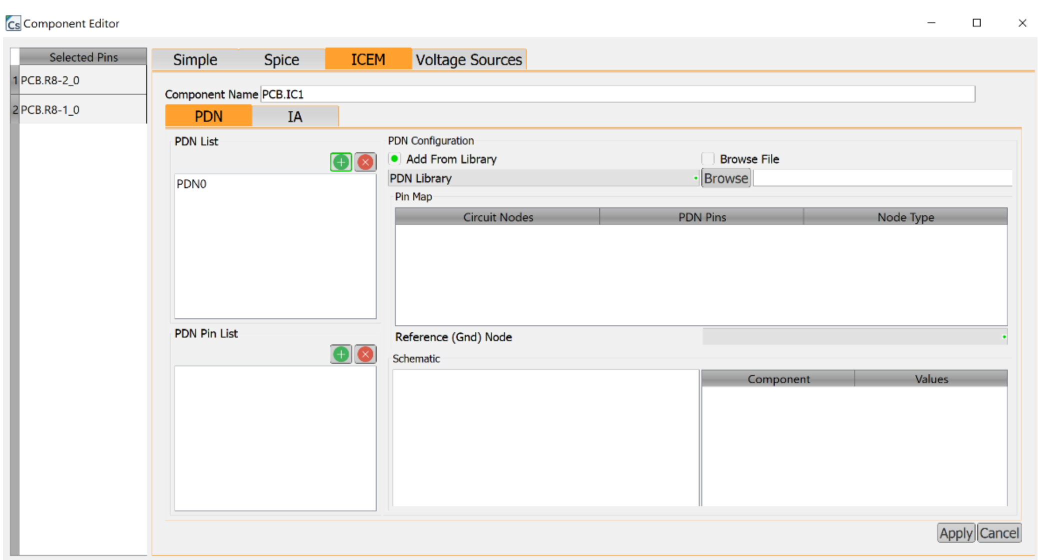


Figure 7-22: Add PDN

(b) Select pins for each PDN

Pins can be populated for each PDN by selecting the PDN (e.g. PDN0) and clicking the “+” button on the “PDN Pin List”. Then select the appropriate pins from the entire pin list and click OK.

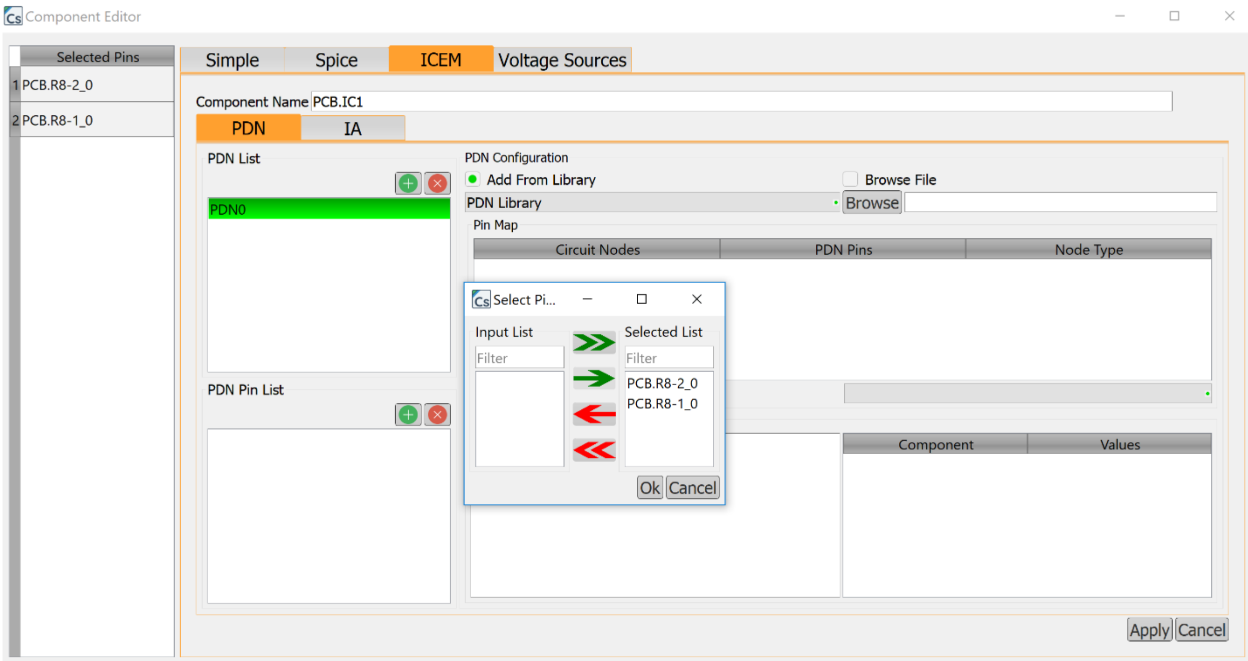


Figure 7-23: Adding pins to each PDN

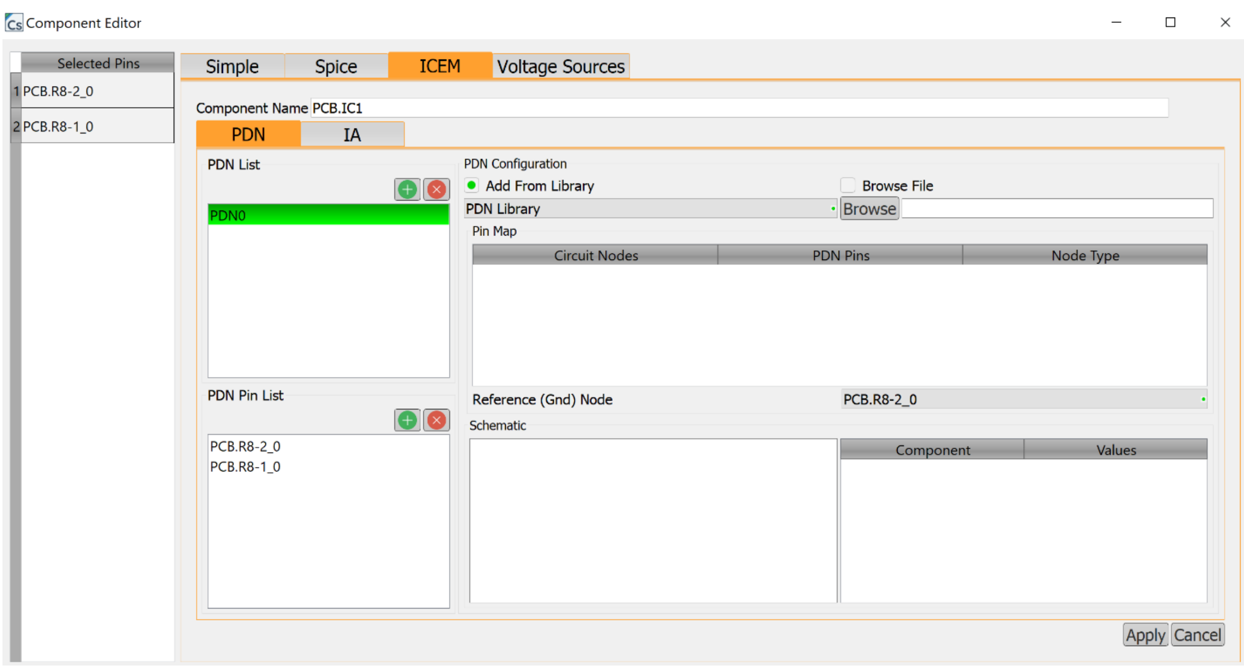


Figure 7-24: Pins in the PDN pin-list

(c) A PDN can be characterized by a SPICE or a Touchstone file using the Browse option

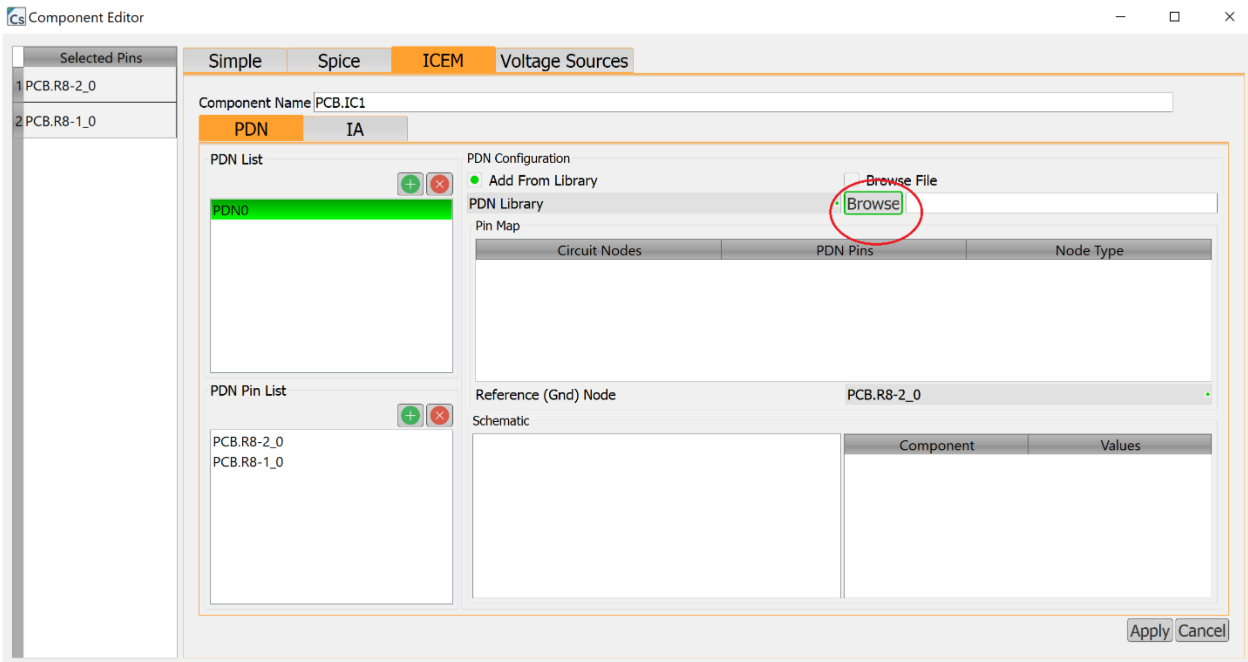


Figure 7-25: Selecting a SPICE or Touchstone file to populate the PDN

(d) A PDN can also be defined using the library option. Currently the library supports only a tank circuit.

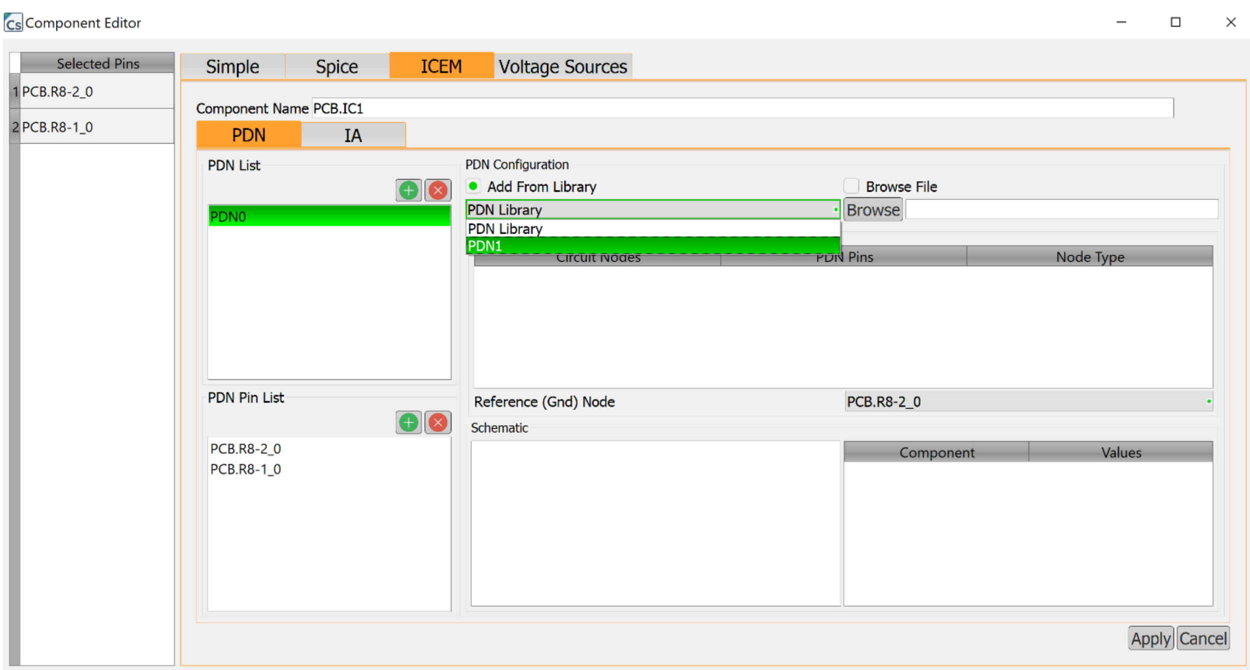


Figure 7-26: Selecting a PDN from the library

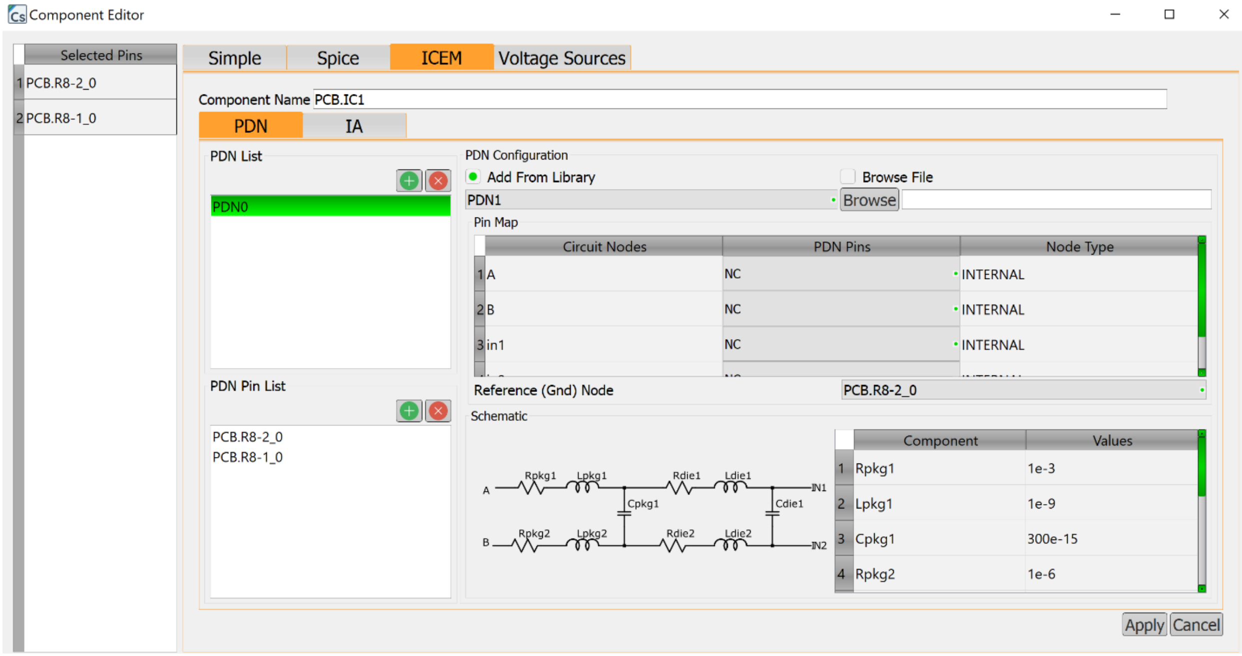


Figure 7-27: Library tank circuit for PDN

(e) Assigning pin connections: some of the pins of the PDN correspond to PCB pins (EXTERNAL) and some to chip pins where the IA is connected (INTERNAL). Only the EXTERNAL or PCB pins need to be mapped at this stage.

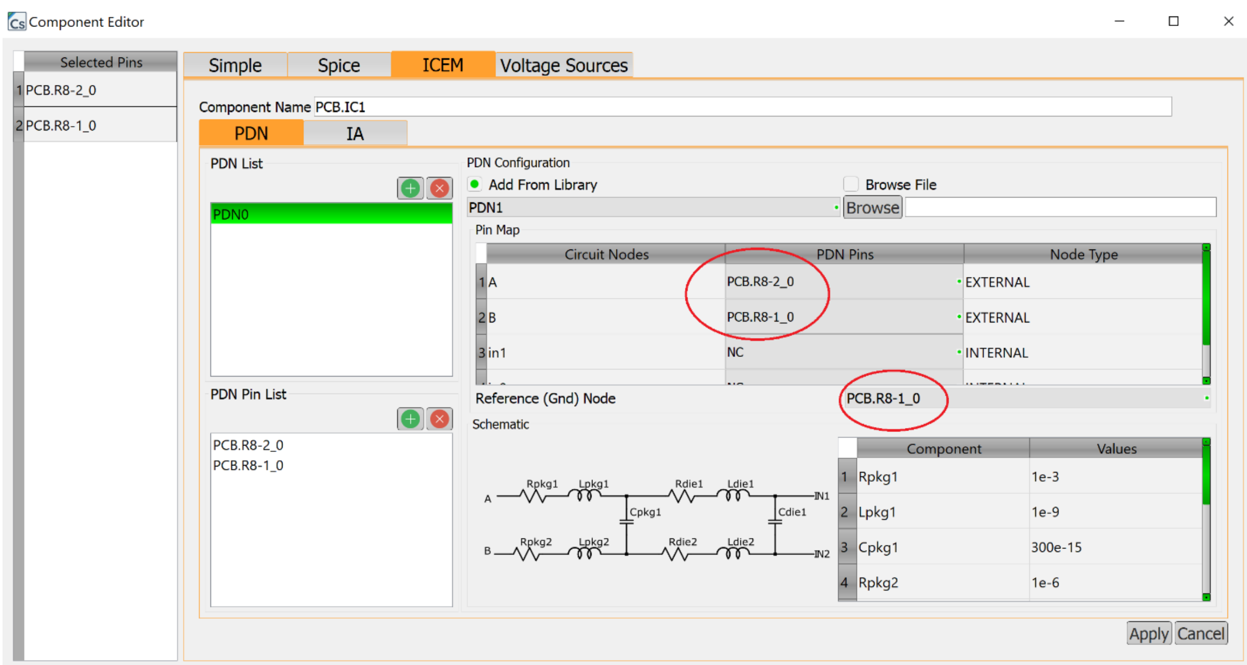


Figure 7-28: Assigning PCB pins to ICEM PDN schematic node-names

(f) Click on “Apply”

7.3.2.2 IA

(a) Step 1: In the IA tab, click on the PDN where the IA is desired. Click the “+” on the IA List to generate a new IA.

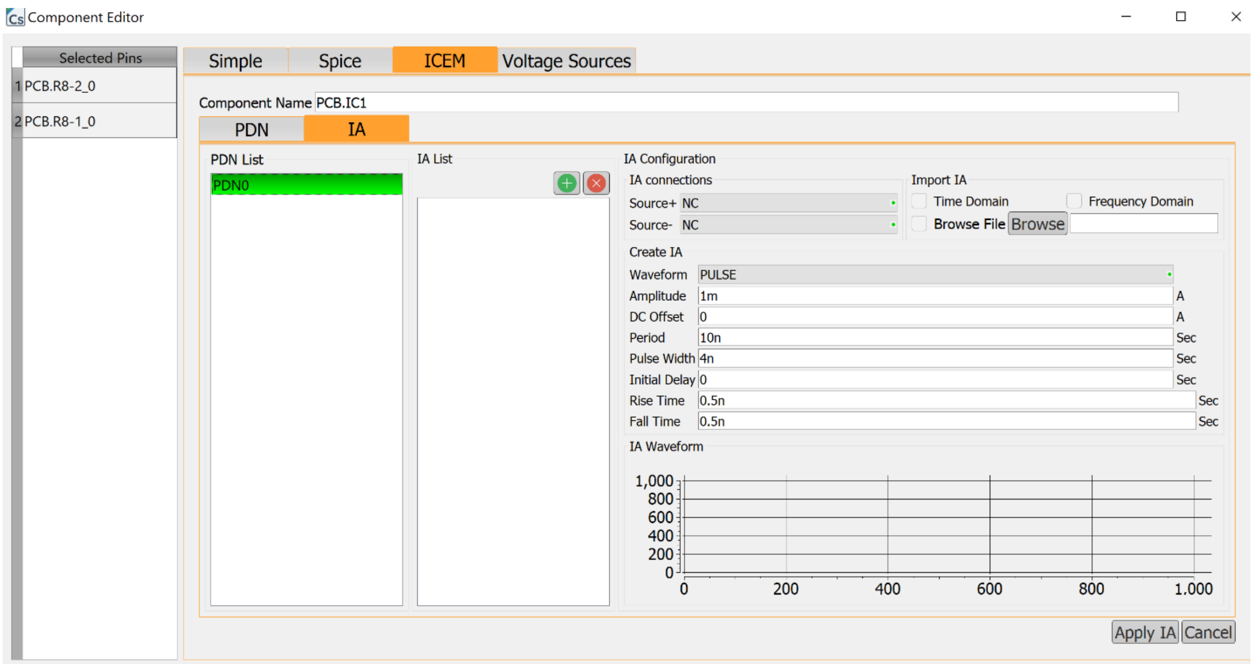


Figure 7-29: Select the PDN to apply the IA

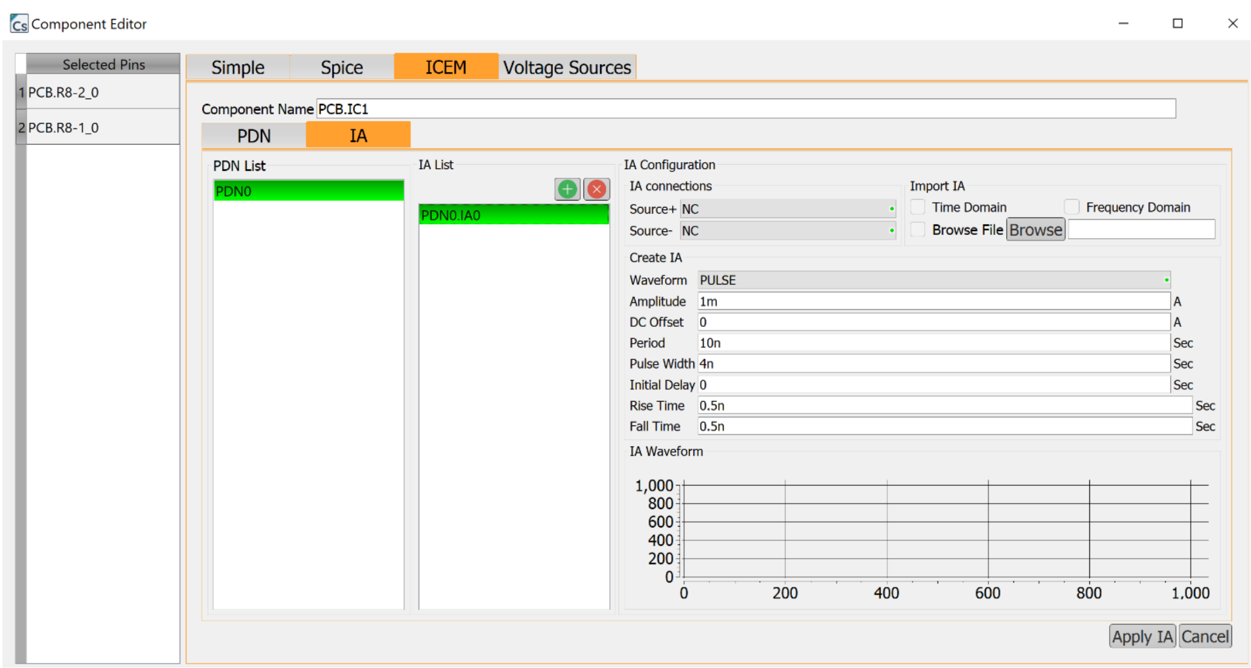


Figure 7-30: Add a new IA to characterize

The IA is a time or frequency domain noise waveform. It can be imported from a csv file by selecting “Time Domain” or “Frequency Domain”. Or it can be loaded using standard waveforms like PULSE etc. from the GUI.

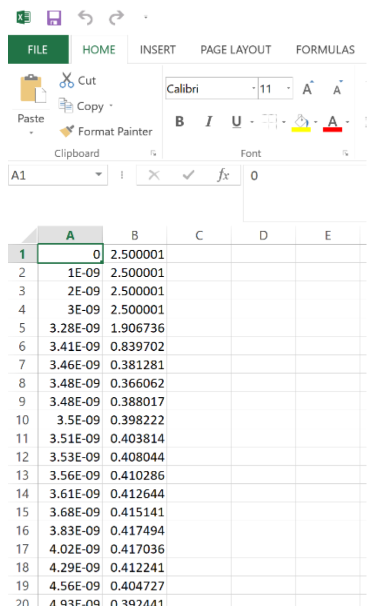


Figure 7-31: Example of a time-domain noise profile file

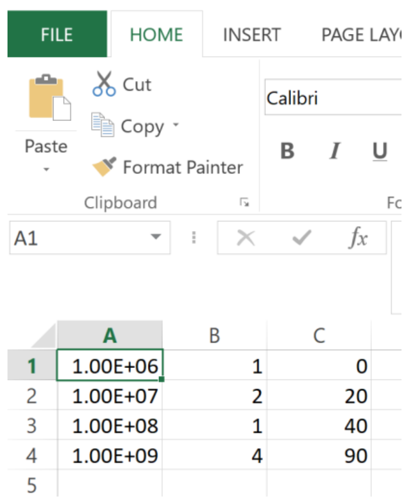


Figure 7-32: Example of a frequency domain noise profile with frequency, noise amplitude and noise phase in degrees

(b) Select the pins across which the noise is injected. The pins that can be selected are the INTERNAL pins of the PDN.

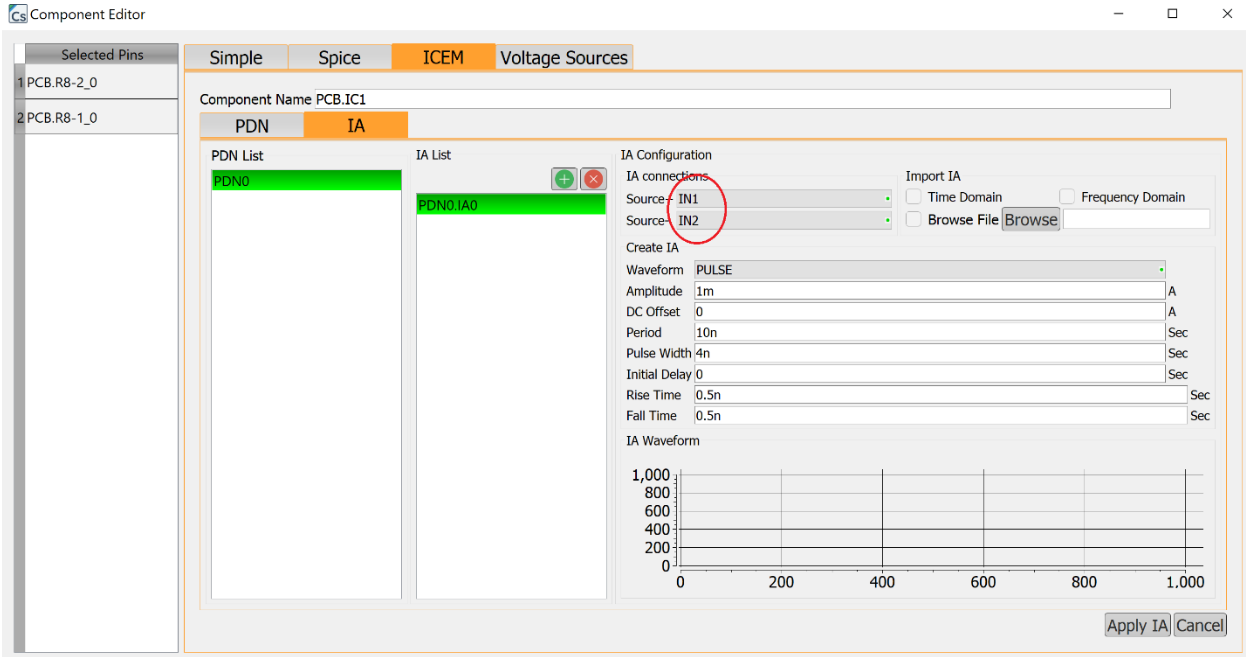


Figure 7-33: Selecting the source+ and source- pins for the noise

(c) Either import the noise file in time or frequency domain by using the browse option OR use the template time domain noise from the GUI. Click on "Apply IA" to visualize the waveform.

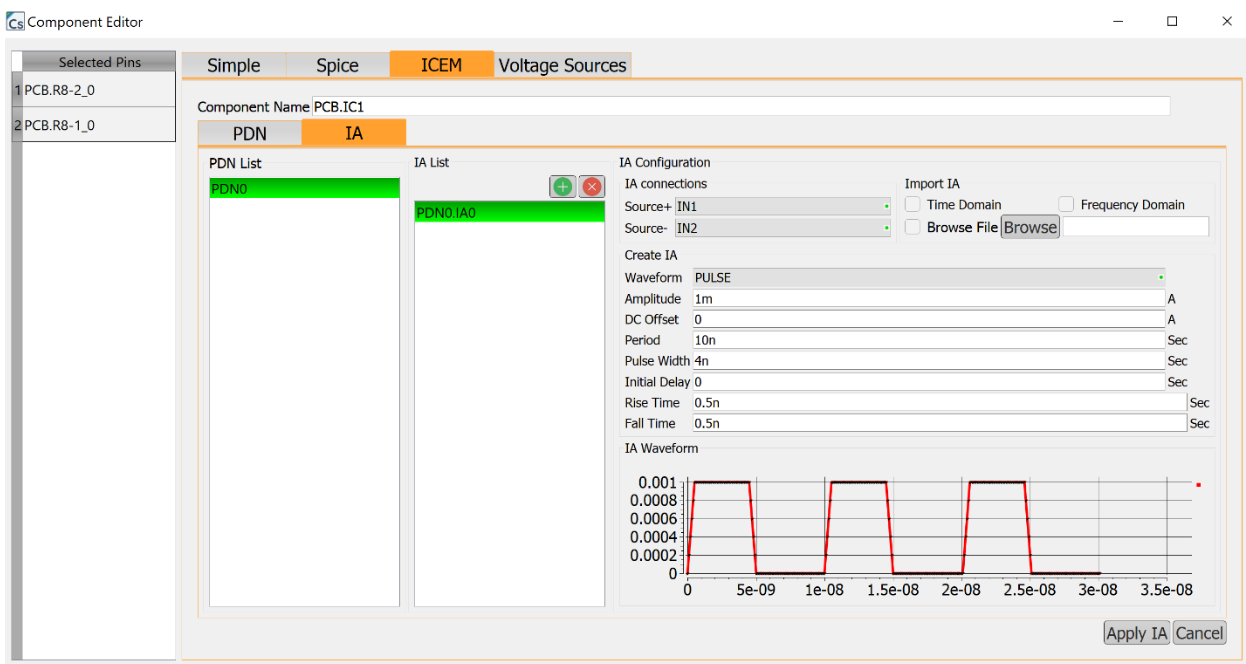


Figure 7-34: Visualization of the IA

8 Chapter 8: Apparatus Interfaces (wiring between apparatuses)

In this chapter, the interface between two apparatuses is described. For a CompScope BCI experiment, the possible combination of interfaces that can result from different apparatus settings are: cable-loadbox interface, cable-connector interface, PCB-connector interface and PCB-housing interface, PCB-cable interface (if connector is deleted). The link between cable and both the clamps, i.e. cable-clamp interfaces are already taken care of internally inside the tool (see 13.3.1 Explanation of CompScope injection clamp library model), and the user need not define those.

8.1 Cable-Loadbox Interface

This interface is used to connect the cable and the Load-box. The user can setup the connections in this apparatus interface once the cable and the loadbox apparatus models have been defined. If there are N cables in the cable Harness then the cable nodes are numbered from n0plus to nN-1plus.

The load-box nodes can be of two types:

(a) Load-box as SPICE subckt file: The load-box nodes are the same as the node names in the .subckt definition, as shown in Figure 8-1 (for a 2 cable system):

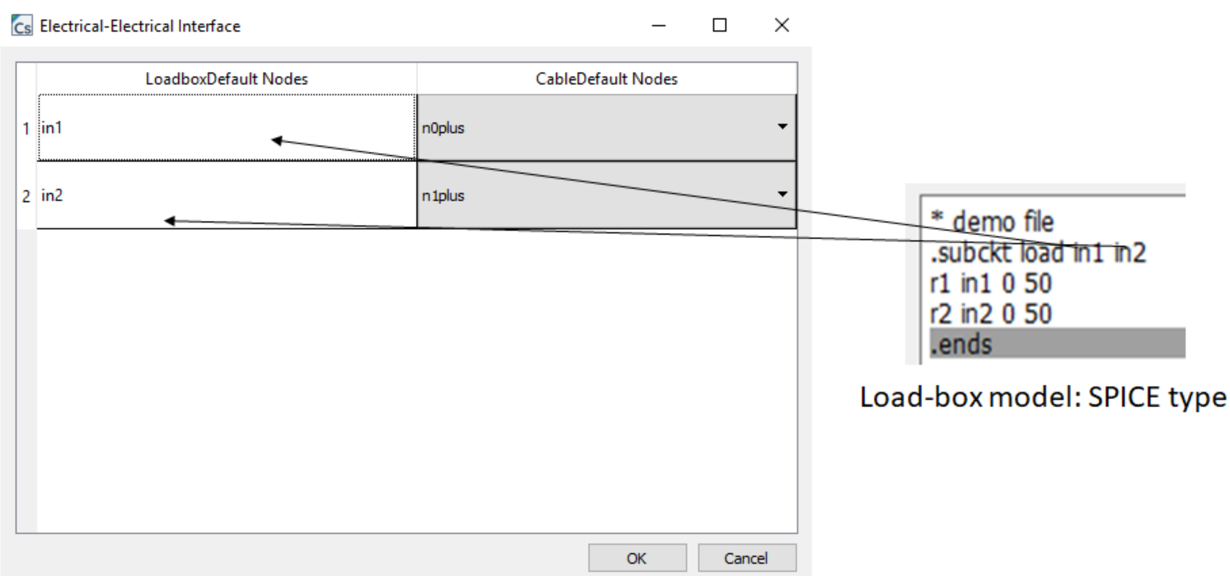


Figure 8-1: Cable-Loadbox interface for SPICE model Loadbox

(b) Load-box as SNP file (touchstone data): The load-box nodes are numbered successively from port1 to portN, where N is the number of ports, as shown in Figure 8-2.

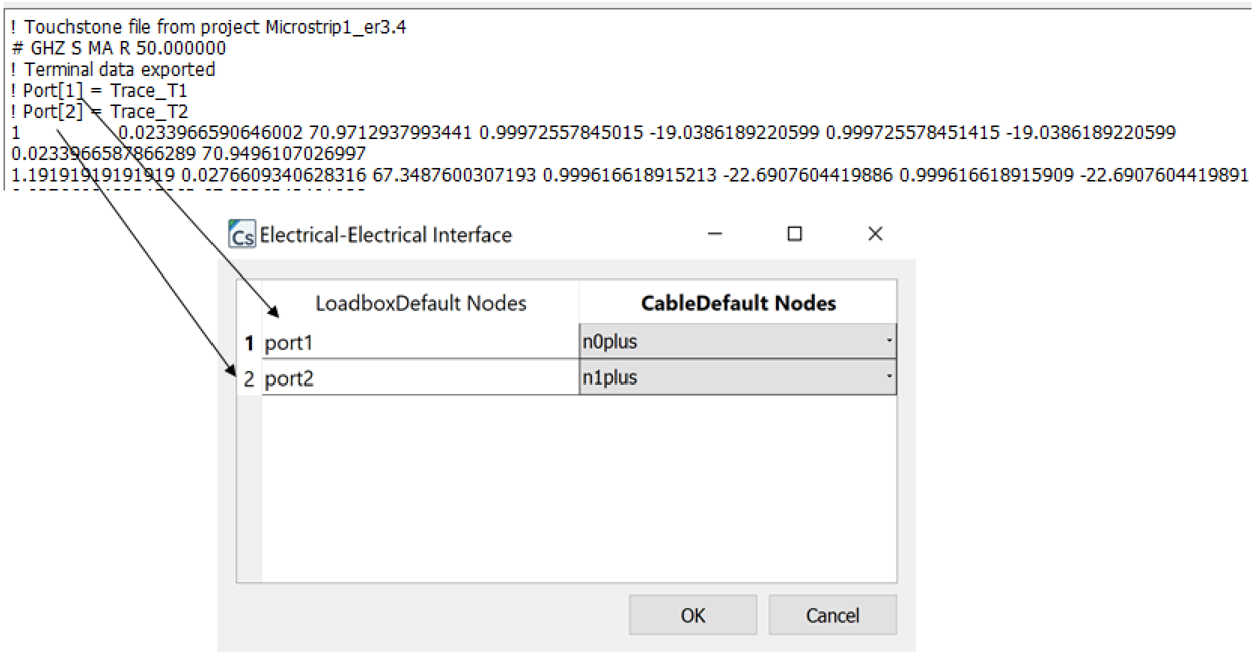


Figure 8-2: Cable-Loadbox interface for Touchstone file Loadbox model

8.2 Connector-PCB Interface

8.2.1 Pre-requisites:

- Import a PCB model and finalize it (chapter 0)
- Import a Connector model and finalize it (chapter 0)

8.2.2 Merge Operation:

Merge is the name of the operation given to creating an interface between two physical models (here Connector and PCB). The step-by-step method of Connector-PCB merge is explained below:

Step 1: Once the finalize operation has been called on PCB and Connector, expand the Apparatus List window. A “Connector_PCB_Interface” appears under both “ConnectorDefault” and “PcbDefault” apparatus. Double click on either of them and a new window called “P-P Interface” opens up as shown.

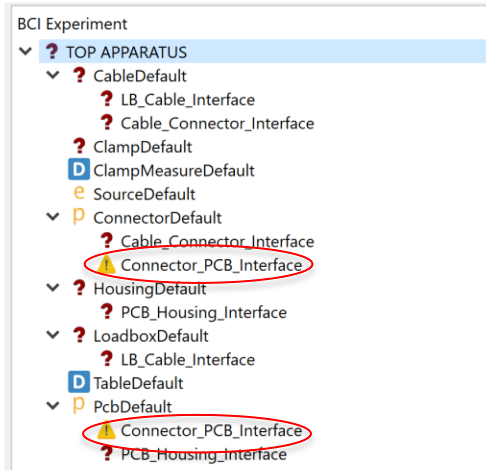


Figure 8-3: Connector-PCB interface

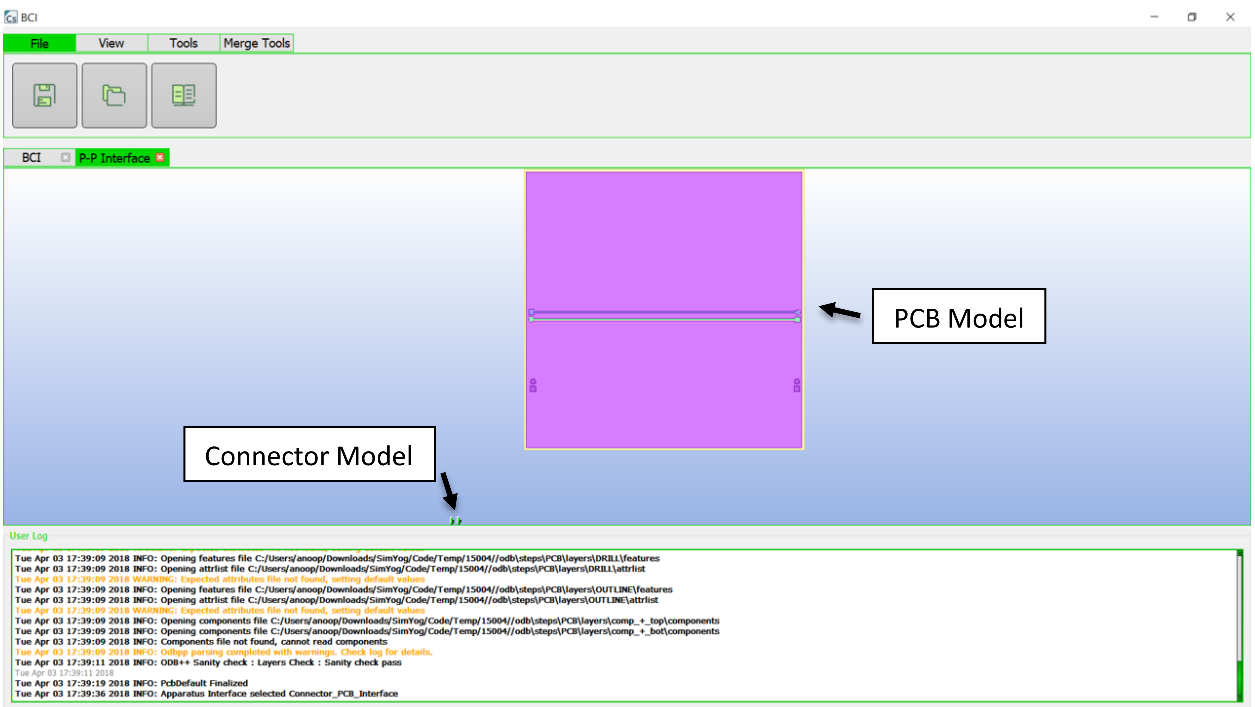


Figure 8-4 PP Interface Window

Step 2: In the PP-Interface you will now see both the Connector and the PCB Model. Select 'View Option' and rotate the view so that both models are visible. Now click on Merge Tools > Translate model to translate the Connector Model. Note that only the connector can be translated, and the PCB remains fixed. Translate using the three-axis translation tool that appears at the connector and bring the connector close to the PCB as shown. Similarly, the model can be rotated using the three-axis rotate tool. Hence use the 3 pin align tool to align the connector and PCB. Refer to the manual video (linked below) to learn about the usage of the tool. You may make rotate, translate and alignment tools as many times as you wish, to reach optimum alignment.

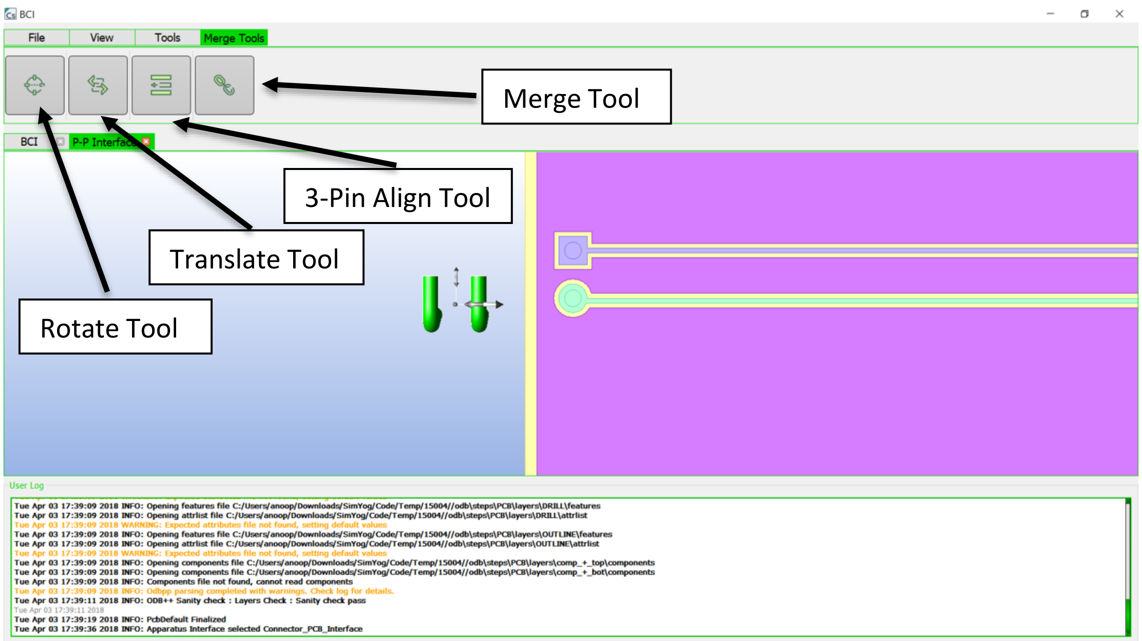


Figure 8-5: Translate and Rotate Connector Model using three-axis translation and rotation tool

Step 3: Click "Merge" button when the desired alignment between the connector and PCB is reached. Now the "PP-Interface" can be closed. Note that there should be minimal penetration of the connector into the PCB. That is, only the bodies of the connector intended to intersect the PCB must intersect with the corresponding nets on the PCB.

Step 4: Note that in the Apparatus List window a new apparatus is displayed – "MergedApparatus". Double click on "MergedApparatus". The 3D Visualization Window shows the connectors merged with the PCB traces and thus are displayed in same color after merge.

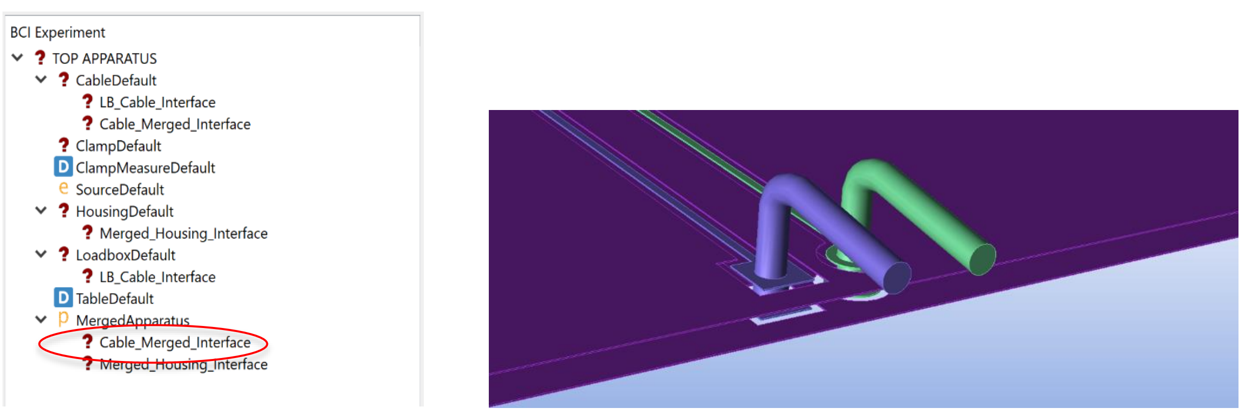


Figure 8-6: Merged Apparatus: Connector-PCB

8.2.3 Video Demonstration:

The following video demonstrates the merge operation in detail

http://simyog.com/resources/manual_animations/ConnectorMerge.mp4

Also, the video in the example 11.5 demonstrates merge operation in more details.

8.3 Housing-PCB Interface

The Housing-PCB interface helps the users define the connections between Housing Apparatus and PCB Apparatus, and follows a similar process as the Connector-PCB merge operation explained above in 8.2. The users are thus suggested to refer to that section for step-by-step details. In addition, the video in the example 11.5 demonstrates the process in further details.

8.4 Cable-Connector Interface

8.4.1 Pre-requisites:

- Import a connector model (chapter 0)
- Construct the cable and perform relevant operations (chapter 6.1)

8.4.2 Adding pins:

One end of the connector connects to the PCB (8.2) while the other end connects to the cables. Hence pins are required to be drawn on the side of the connector that needs to be connected to the cables. The users are suggested to draw that many number of pins as the number of cables that are required to be connected to the connector. Pins on the connector are added as explained in **Error! Reference source not found.**.

8.4.3 Defining Cable-Connector Connections:

This operation has been refrained from being called as “merge” since this is a connection between electrical (cable) to physical (connector) model.

Step 1: Once the cables are created and finalize operation is called on Connectors with pins defined, expand the Apparatus List window. Please note that “Cable_Connector_Interface” appears under both “ConnectorDefault” and “CableDefault” apparatus. Double click on either of them to “Open Apparatus”. The Cable-Connector interface opens up as shown.

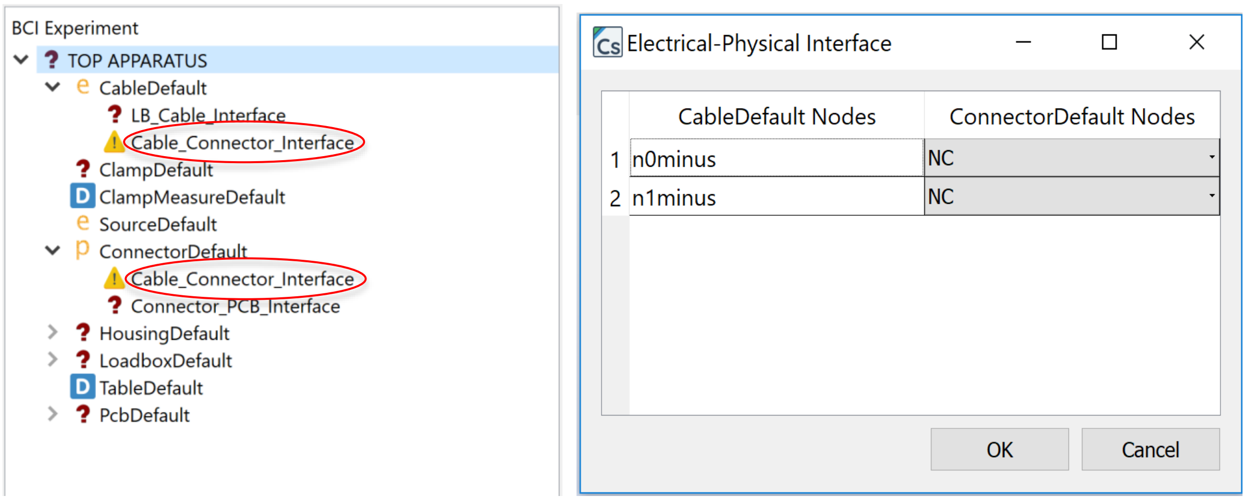


Figure 8-7: Cable-Connector Interface

For the example shown, two cables are made a priori. The cable nodes are named as n0minus and n1minus.

Step 2: Scroll through the pins under the column "ConnectorDefaultNodes" and select the appropriate Connector pin that needs to be aligned to the corresponding Cable node. For this example, the pin naming and placement has been such that cable node n0minus aligns with pin p1 of connector and cable node n1minus aligns with pin p2 of connector, for a proper connection.

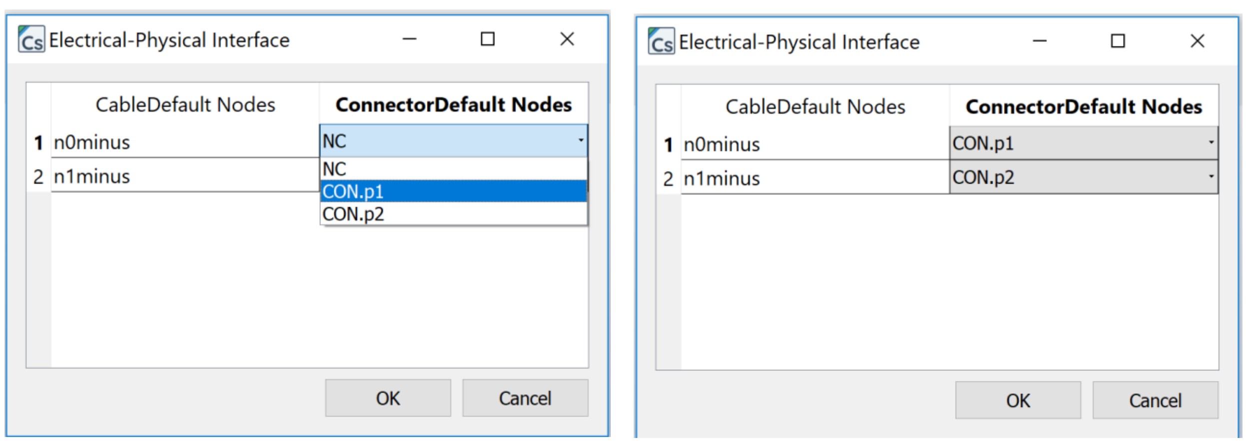


Figure 8-8: Cable nodes - connector pins alignment

Step 3: Click "OK" when connections are complete as desired.

Note that the connections made are displayed using tick signs.

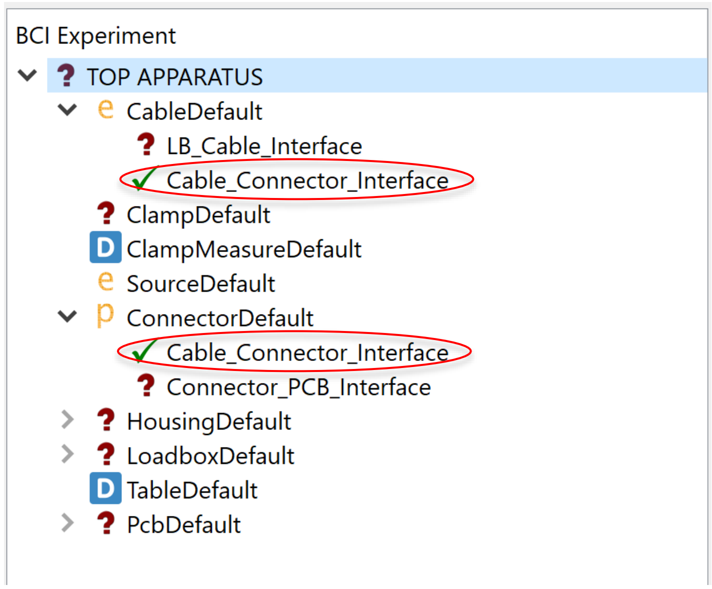


Figure 8-9: Cable-connector interface connections complete

8.5 Cable-PCB Interface

Cable-PCB interface is generated only when the user deletes beforehand the connector interface for simplicity and wishes to connect the cables directly to the pins on the PCB.

8.5.1 Pre-requisites:

- This configuration is only possible when the connector and housing apparatuses have been deleted
- Import a PCB model (chapter 0)
- Construct a cable model and perform relevant operations (chapter 6.1)

8.5.2 Adding pins:

Pins are to be added at appropriate locations on the PCB where the cables are desired to be connected. The users are advised to refer to 6.10.5 for adding pins to PCB.

8.5.3 Defining Cable-PCB Connections:

Step 1: Once the cables are constructed and finalize operation and pin addition is done on the PCB, expand the Apparatus List window. Please note that “Cable_PCB_Interface” appears under both “CableDefault” and “PcbDefault” apparatus. Double click on either of them to “Open Apparatus”. The Cable-PCB interface opens up as shown.

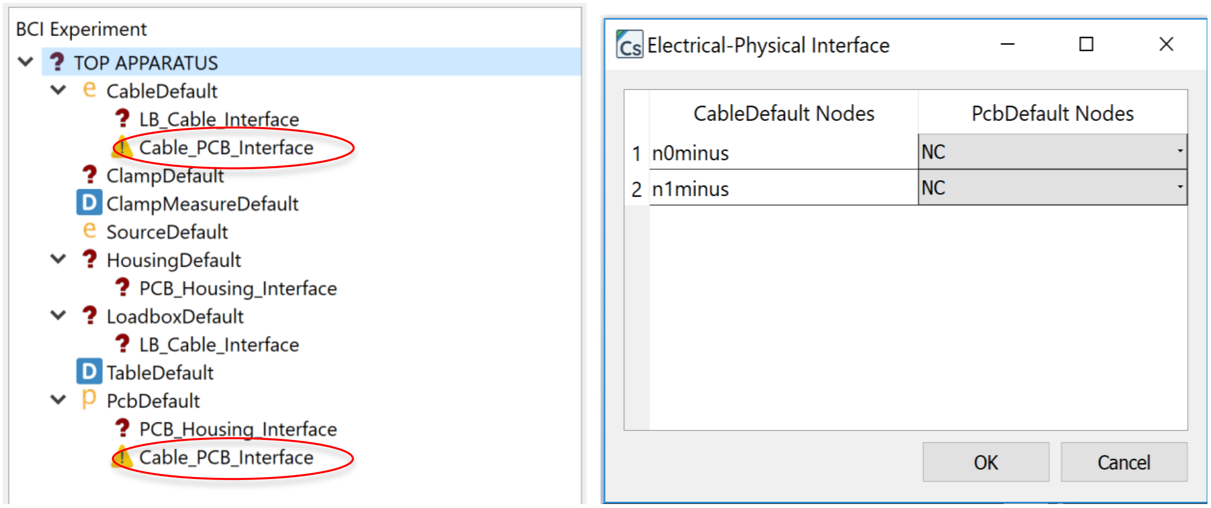


Figure 8-10: Cable-PCB Interface

For the example shown, two cables are made a priori. The cable nodes are named as n0minus and n1minus.

Step 2: Scroll through the pins under the column “PcbDefaultNodes” and select the appropriate PCB pin that needs to be aligned to the corresponding Cable node. For this example, the pin naming and placement has been such that cable node n0minus aligns with pin p1 of PCB and cable node n1minus aligns with pin p2 of PCB, for a proper connection.

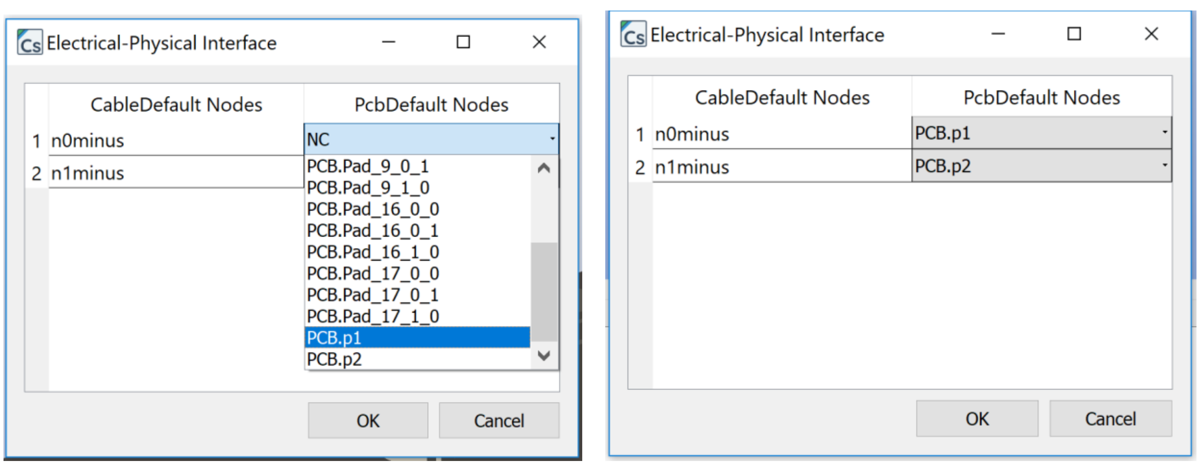


Figure 8-11: Cable nodes - PCB pins alignment

Step 3: Click “OK” when connections are complete as desired.

Note that the connections made are displayed using tick signs.

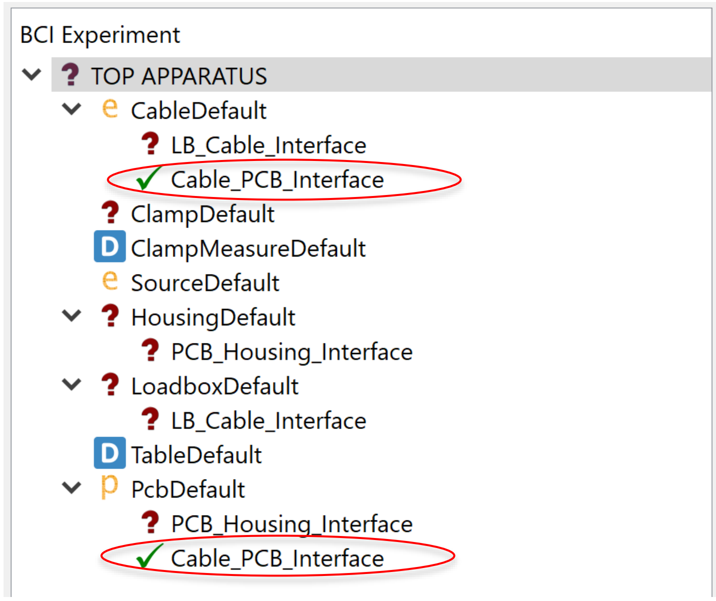


Figure 8-12: Cable-PCB interface connections complete

9 Chapter 9: Operations

In this chapter, some of the common operations are reviewed.

9.1 Finalize

The “Finalize” operation applies to any 3D geometry imported. This holds for PCB, connector or housing apparatuses.

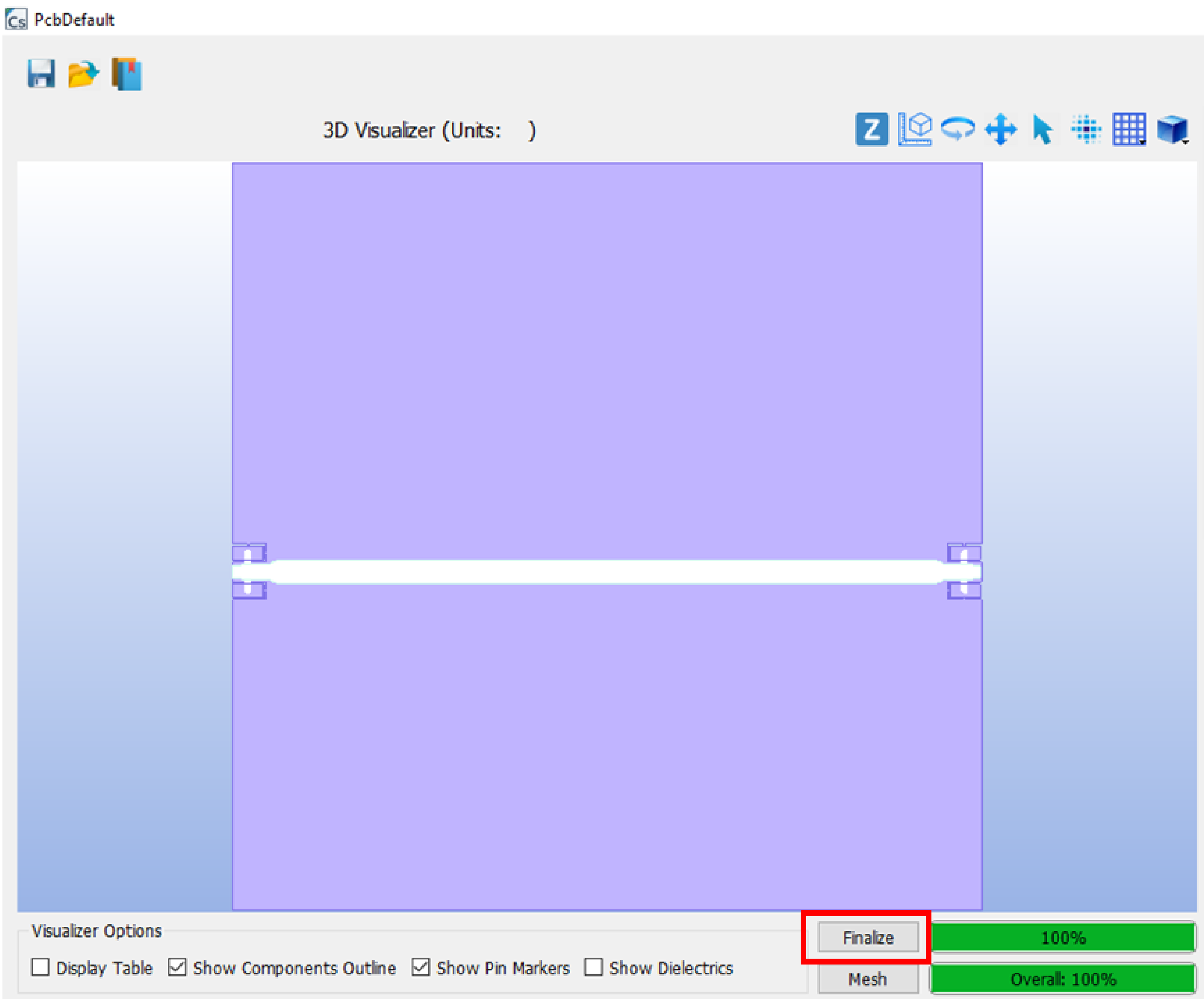


Figure 9-1: The "Finalize" button in the PCB apparatus

During the finalize operation, some of the geometric processing of the imported CAD module occurs, e.g. (a) uniting touching nets (b) assigning material properties to 3D object surfaces in preparation for surface meshing etc.

The “Finalize” operation must be performed before any merge operation or before the final Analyze step. If the user forgets to press “Finalize”, CompScope for most cases will automatically “Finalize” before merge or “Analyze”.

Some operations that alters the geometric properties of the structure will mark the structure as "To be Finalized". If there is no To-Be-Finalized geometry the "Finalize" button click will do nothing, as shown in log messages as **INFO: Nothing to finalize** .

9.2 Mesh

The "Mesh" button performs a surface-meshing of the structure to be analyzed.

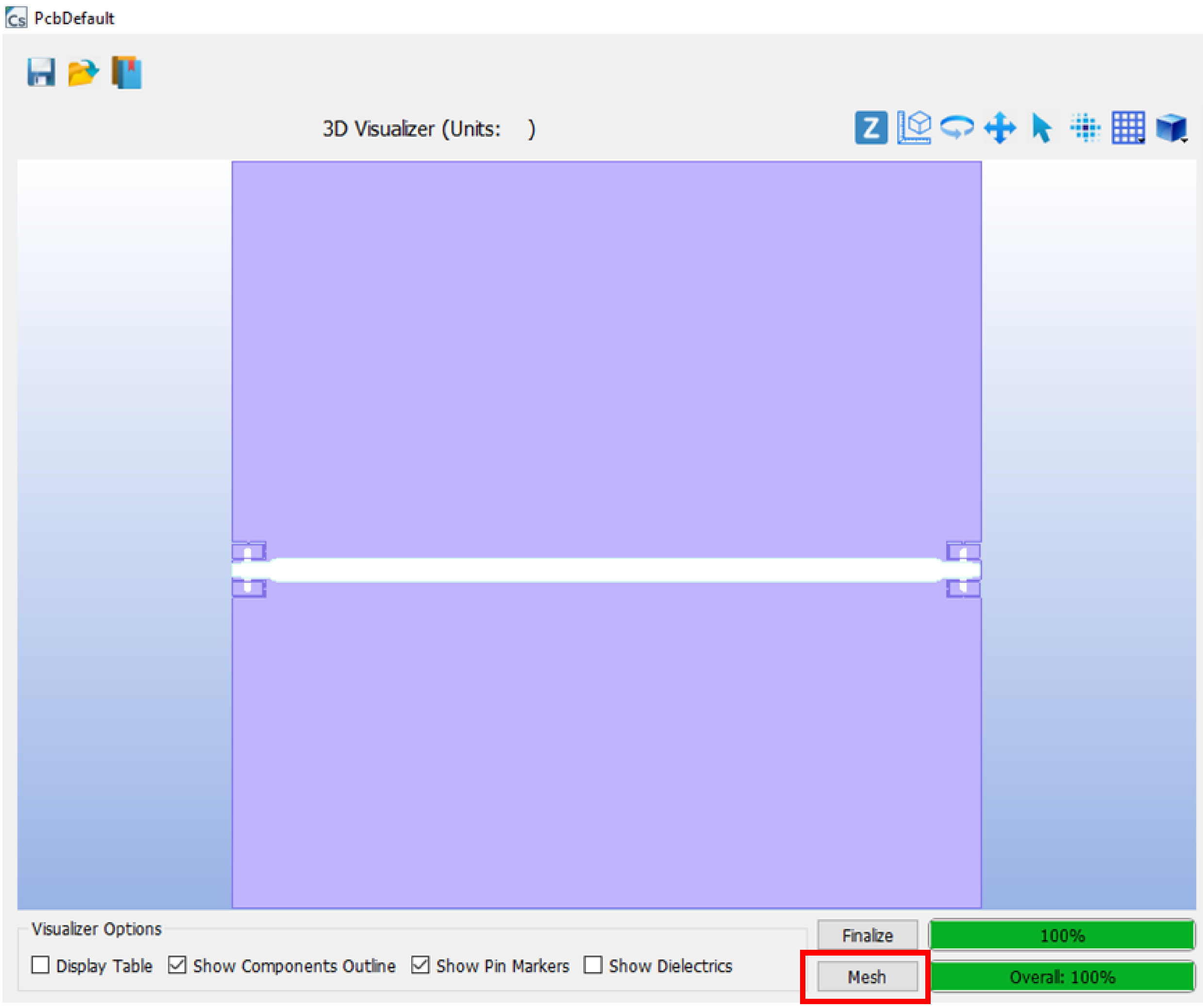


Figure 9-2: The "Mesh" button on the PCB apparatus

CompScope uses a surface-mesh formulation. This means that the surface of conductors and the dielectric interfaces will be meshed using planar elements, in most cases triangles.

Although it is theoretically possible to "Mesh" the apparatus at any stage, CompScope prevents meshing an apparatus unless the experiment setup is completed, including configuring the apparatus interfaces. Such a log message is shown below:

PcbDefaultERROR: Mesh INVALID FUNCTION CALL: Experiment needs to be completed before mesh

This is because, meshing is sometimes time consuming, and there is no need to be meshing a PCB if it is going to be merged later with connector/housing. Therefore, CompScope only allows the mesh process after the complete experiment setup.

Just like finalize, if the user does not press “Mesh”, it will automatically be called when the “Analyze” button is pressed. Also, meshing requires the structure to be “Finalized”.

9.3 Analyze

The “Analyze” button triggers the system solution.

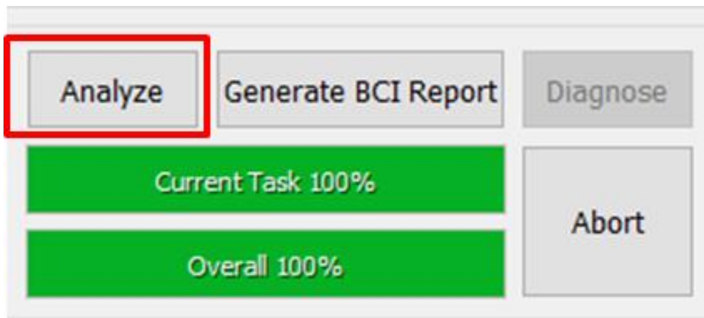


Figure 9-3: The Analyze button

The “Analyze” operation will fail if the experiment setup is incomplete, which can happen in any of the two following situations:

(a) any of the apparatus model is missing

(b) any of the apparatus interface connection is missing

(c) the solution frequency details have not been set

9.4 Generate Report

The “Generate Report” button will produce a PASS/FAIL excel sheet after the “Analyze” operation is complete.

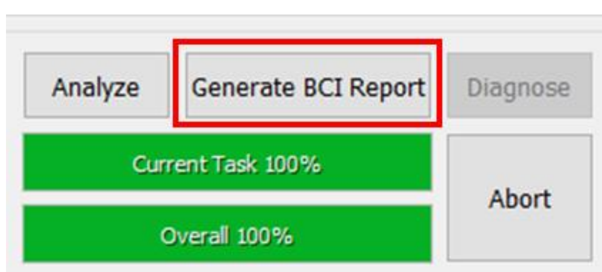


Figure 9-4: Generate BCI report button

The "Generate Report" will only generate the pass-fail excel sheets if:

(a) There was at least 1 ICIM component

(b) There was at least 1 IB property specified in the ICIM components

(c) Harmonic Balance was used with threshold current/voltage/power specifications of the non-linear elements and subckts.

In any other situation, the "Generate BCI Report" will complain in the log messages as follows:

ERROR : No BCI report available

9.5 Diagnosis

This is a proprietary technology of SimYog and is used to identify the geometries responsible for specific failure conditions. The Diagnosis feature is currently disabled – it will be enabled shortly in a future release.

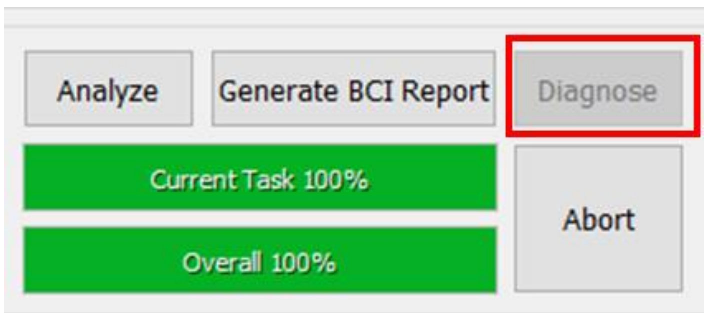


Figure 9-5: The Diagnosis operation button

10 Chapter 10: Results

This chapter deals with result visualization and generating the BCI PASS/FAIL reports.

10.1 Manipulating curve visualization

After the completion of the experiment "Analyze", the results can be viewed from the *Project Tree* in the BCI main window. The Figure below shows one such example.

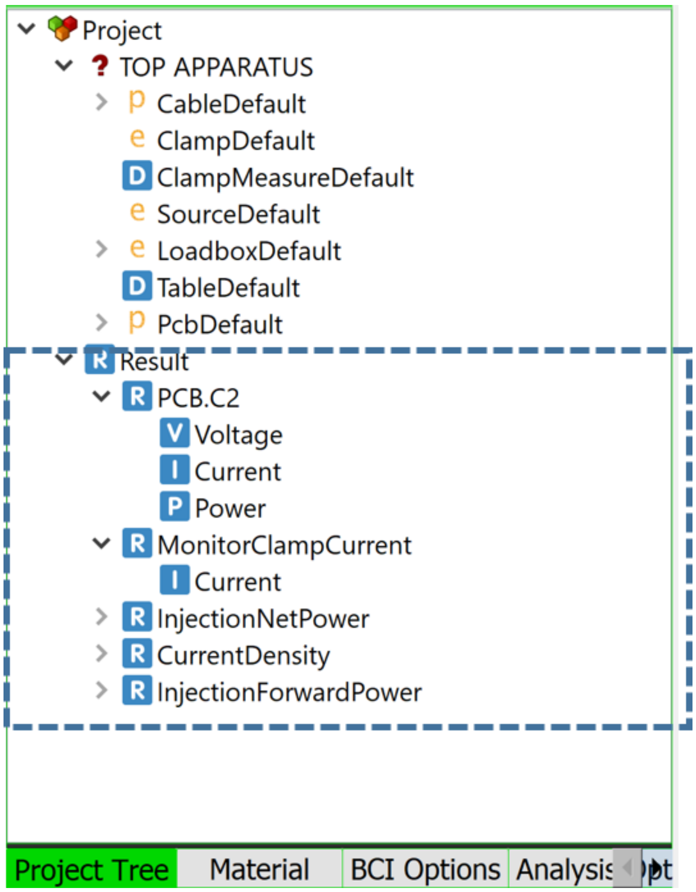


Figure 10-1: Results

The subjects present inside the *Results* are computed in the frequency range and for the points defined under *Analysis Options* in *Options Tab.*

Step 1: Select the subjects to be plotted. On selection, their types and units are displayed in the fields *Parameter to Plot* and *Data Type* respectively. Multiple subjects can be plotted together if their types and units agree.

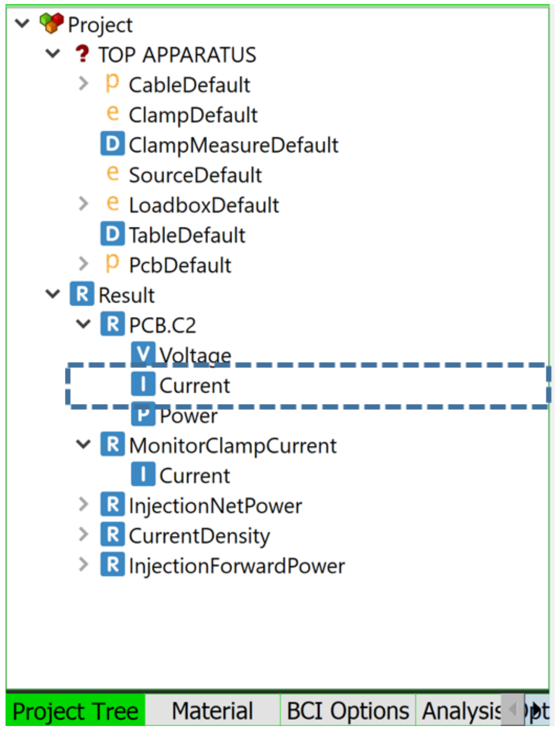


Figure 10-2: Select the quantity to plot

Step 2: The *Data Type* can be changed to change the units of the subject to be plotted.

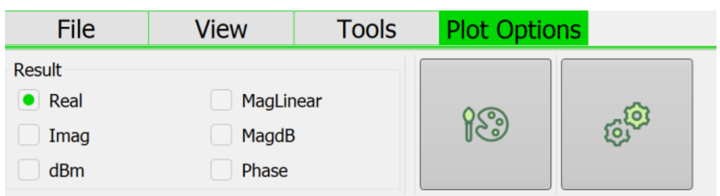


Figure 10-3: Select the parameter type – real/imag/magnitude etc.

Step 3: After selecting the desired subjects and units, click on *Plot.* A new window showing the plot appears which is called the *Result Window.*

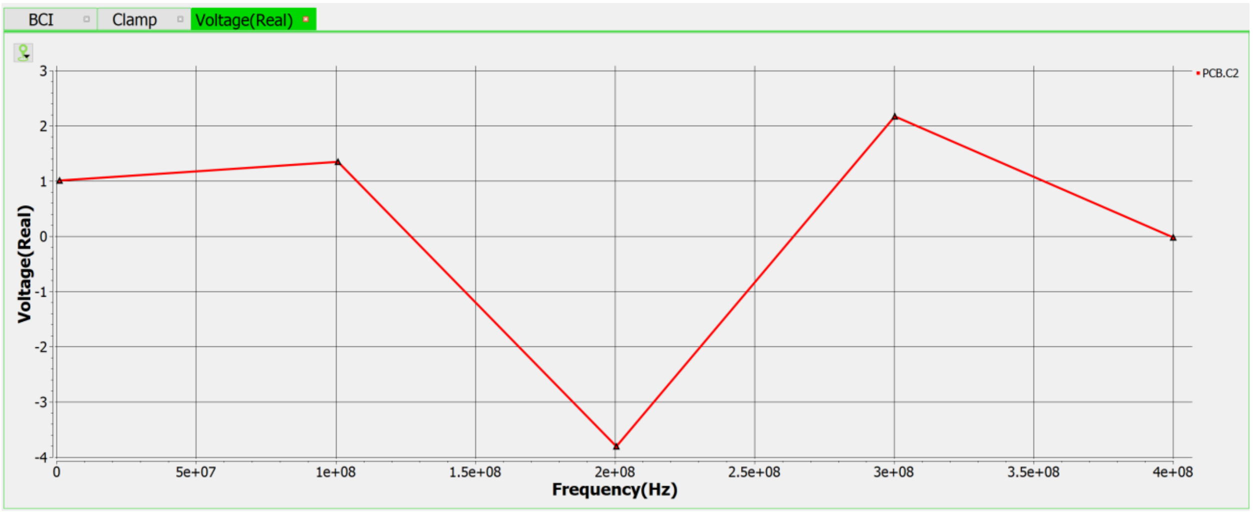


Figure 10-4: Plot visualize.

The curve can be selected by clicking on it. Selection of a curve allows changing its properties such as thickness, marker type and color. Selection of a curve is indicated by the curve color changing to a shade of Green and Yellow.

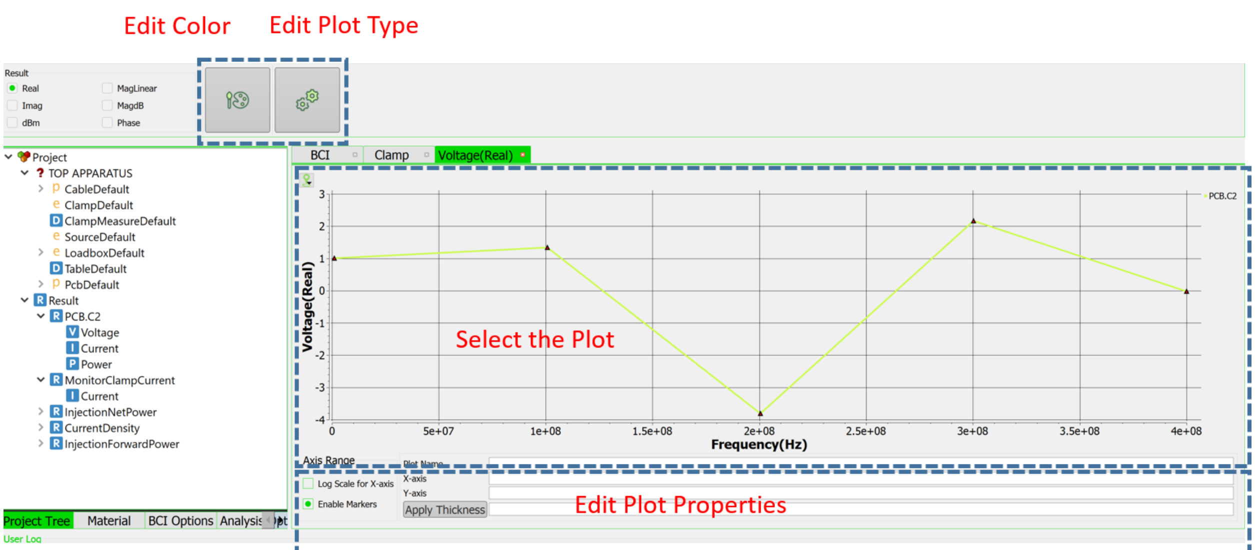


Figure 10-5: Editing Plot properties.

Step 4: The plotted curve/curves can either be exported as a *.csv file or as a *.png file by clicking on the *Save Icon* in the tool bar present at the top of the window.

Figure 10-6: Saving the plot in csv

10.2 BCI results: Component wise Current, Voltage and Power

This section deals with visualizing the results corresponding to the IB curves for the PDN assigned pins.

Step 1: From the *BCI Experiment Main Screen* navigate to the *PCB Default*.

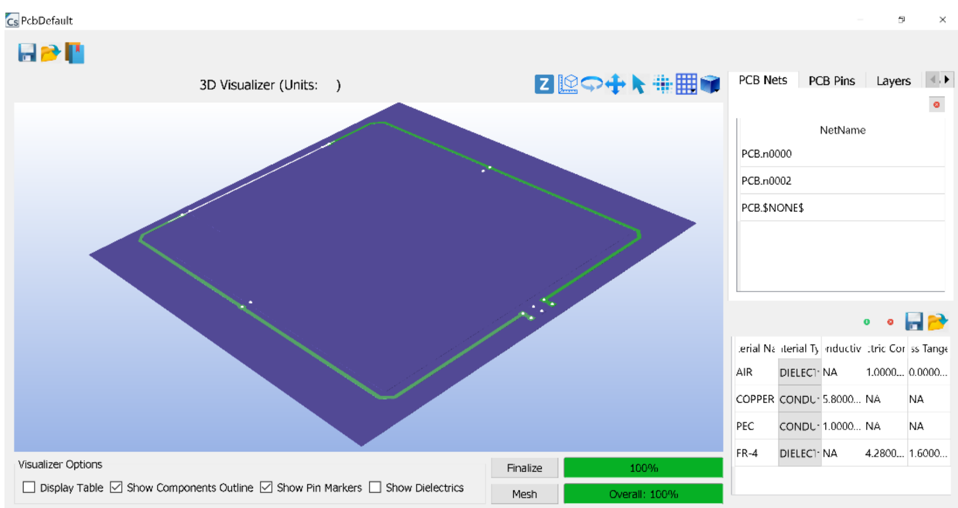


Figure 10-2: PCB window

Step 2: Proceed to the *Components Tab* in *PCB Property Table* and double click on the ICIM Component for which the IB Curve has been defined.

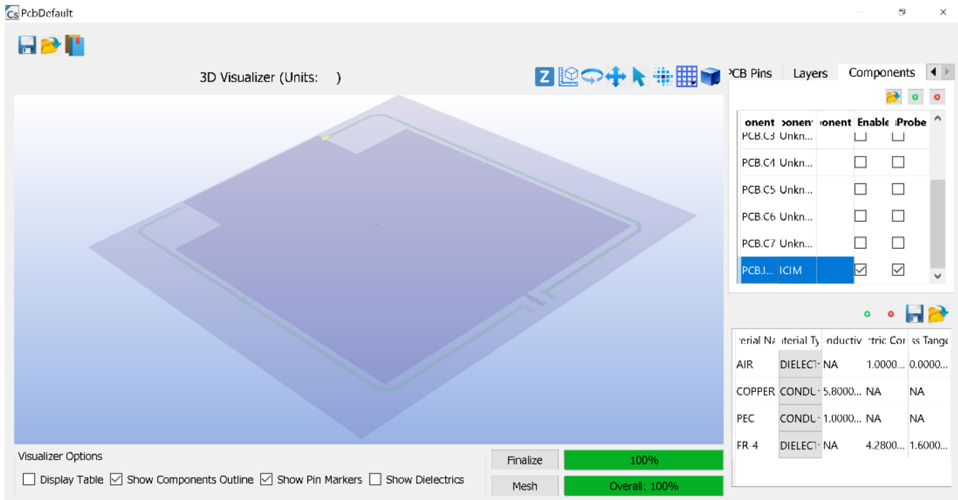


Figure 10-3: ICIM component for which IB had been defined before analysis

Step 3: The action described in *Step 2* opens a new window. Click on the *ICIM Tab* in this window. Under the *ICIM Tab* click on the *IB Tab.*

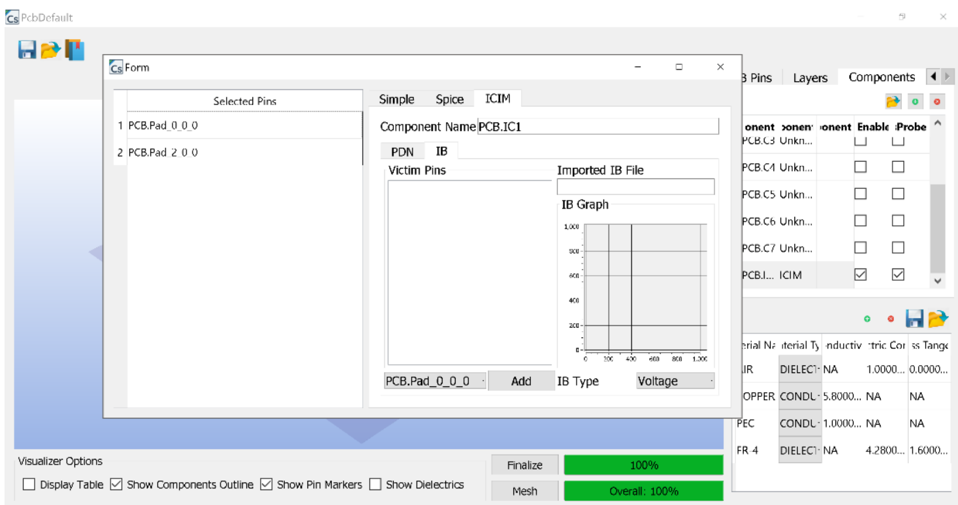
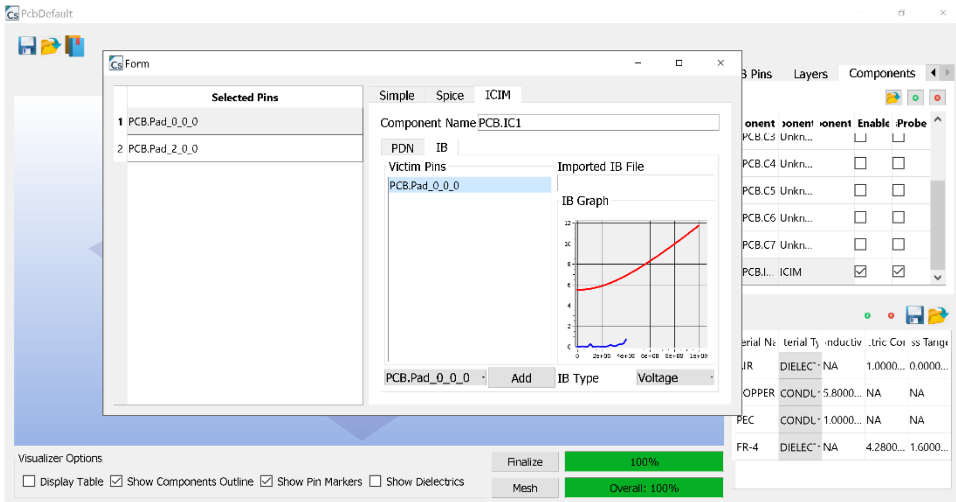


Figure 10-4: IB property window of the ICIM

Step 4: Select **one** of the pins for which the IB (PASS/FAIL) curve is to be viewed from the *Selected Pins* column in the left. Depending on whether a voltage or a power IB limit line was added before the analysis, select the appropriate one from the *IB Type* combo-box. This should populate the *Victim Pins* column on the right.

Step 5: Select a pin from the *Victim Pins* column on the right. If this is performed after analysis of an experiment involving an ICIM component then the *IB Graph* will contain two IB curves. The curve red in color is the limit line which is the threshold for deciding the PASS/FAIL criterion whereas the curve blue in color is the simulated voltage/power at that pin. If the blue curve exceeds the red curve then the report in the previous section displays a FAIL for that pin pair whereas if the blue curve is below the red curve then the report displays a PASS for that pin pair.



10.3 BCI results: Harmonic Balance

When Harmonic Balance is enabled, current/voltage/power waveforms across terminals of non-linear elements or sub-circuits are simulated.

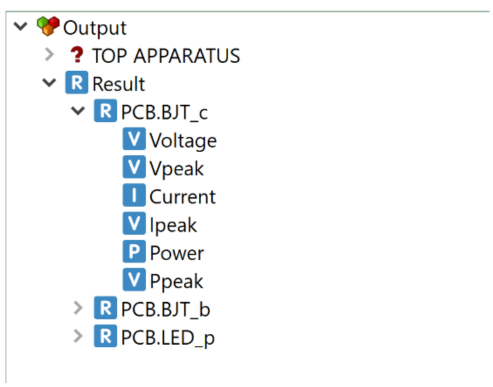


Figure 10-5: Harmonic Balance Results

The “Voltage”, “Current” and “Power” are time-frequency plots.

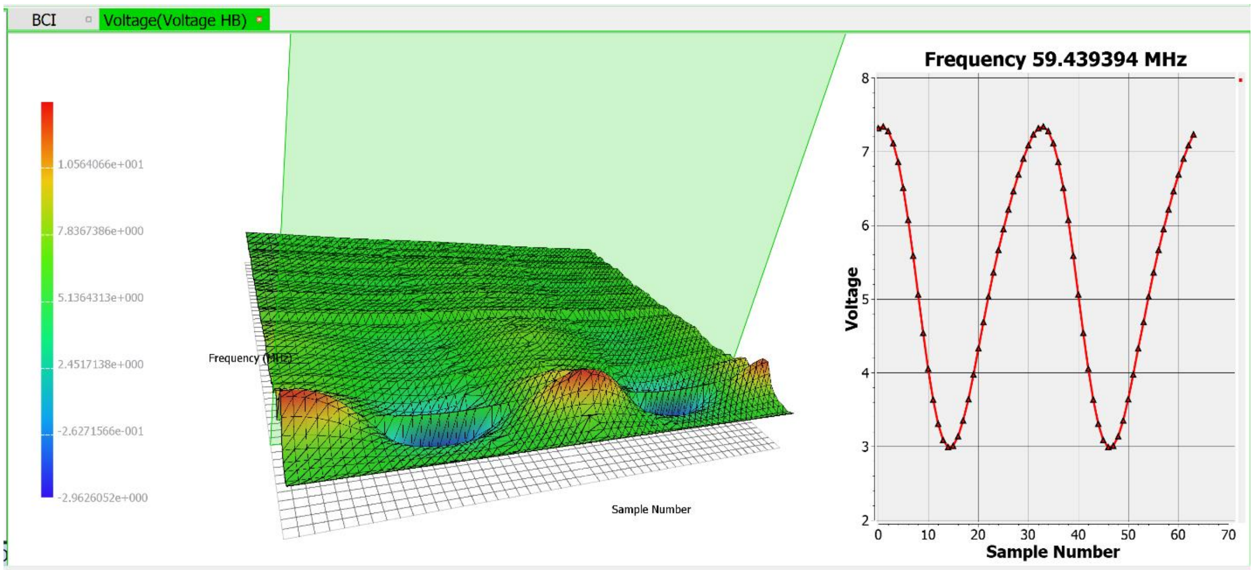


Figure 10-6: Time-Frequency plots for Harmonic Balance Voltage

A frequency cut is also enabled to visualize the waveform at that frequency of BCI noise injection vs. time samples.

The Ipeak/Vpeak/Ppeak are 2D plots vs. frequency showing the peak values of the waveforms across frequencies and can be used to judge pass/fail.

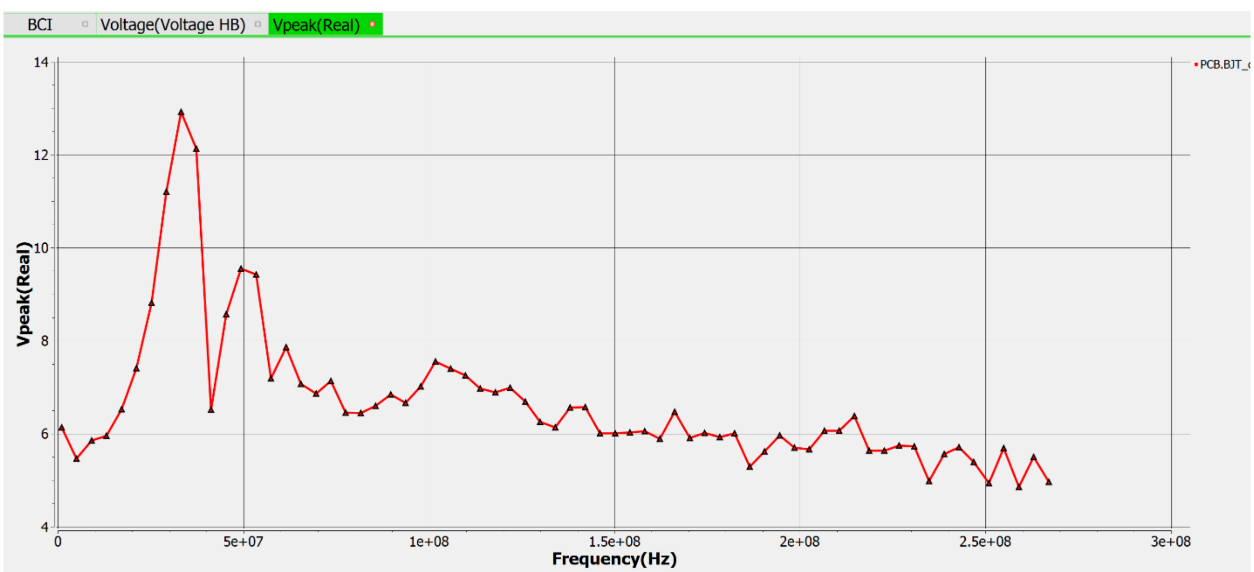


Figure 10-7: 2D peak plots vs frequency useful to judge pass/fail

10.4 Generate Report

This section deals with generating the report. After the "Analysis" is complete, the "Generate Report" button can be clicked to generate an html report with BCI or RE results.

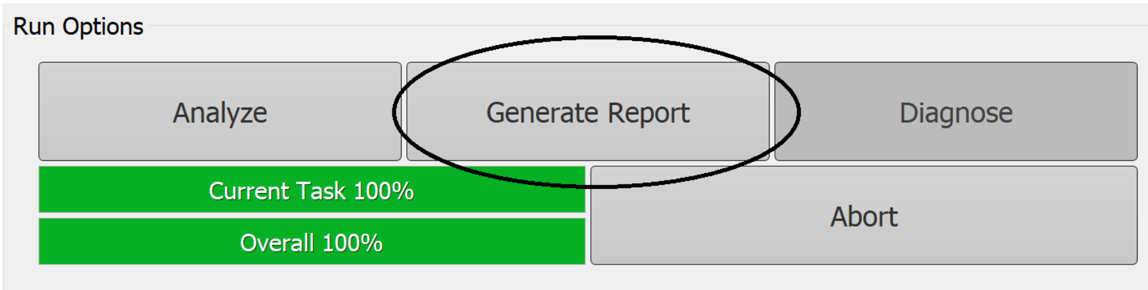


Figure 10-8: Generate report button

The report files are generated in the desired folder.

Figure 10-9: Report files generated

It must be noted that the html file renders the information from the json and png files. Therefore it needs to reside on the same folder and always accompanied by the other 3 files (a) report.json (b) compScope_512.png (c) logo-simyog-gree.png. The html file alone will not show any report.

The report has the following sections:

10.4.1 Header Section

This section demonstrates the header section of the report:

SIMULATION REPORT

Report generated by: Compliance-Scope 2019-01-20.09:46:02

Project	Exp.bci	DUT	PcbDefault
Test	BCI	Date of Analysis	2019-01-20.04:19:59
Start Frequency	1.000000 MHz	End Frequency	400.000000 MHz

Figure 10-10: Header section of report

The date of analysis refers to date and time the project completed solving and the date on the top refers to the date and time the report was generated.

10.4.2 Executive summary

This section presents the key findings from the analysis:

Executive Summary

Component	Susceptible Frequencies	Threshold Values	Simulated Noise Voltages (mV)	Suspect Design
PCB.C149-PCB.C149.1	12 MHz		117.99	PCB
	38 MHz		54.66	PCB
	58 MHz		57.98	PCB
	105 MHz		60.47	PCB
	141 MHz		61.94	PCB
	222 MHz		773.69	PCB
	262 MHz		70.03	PCB
	362 MHz		45.48	PCB
	389 MHz		38.10	PCB
	396 MHz		37.19	Cable
PCB.C521	7 MHz		0.60	PCB
	139 MHz		22.57	PCB
	222 MHz		11.89	PCB
	259 MHz		3.47	Cable
	350 MHz		50.60	PCB
	391 MHz		7.92	PCB
PCB.C522	7 MHz		0.61	PCB
	43 MHz		0.26	PCB

Figure 10-11: Executive summary of BCI experiments

For BCI experiments, the executive summary details the "susceptible frequencies" for the probed components, their threshold values (if present) and the suspect design that caused this high noise.

Executive Summary

E-Field Horizonatal	E-Field Vertical

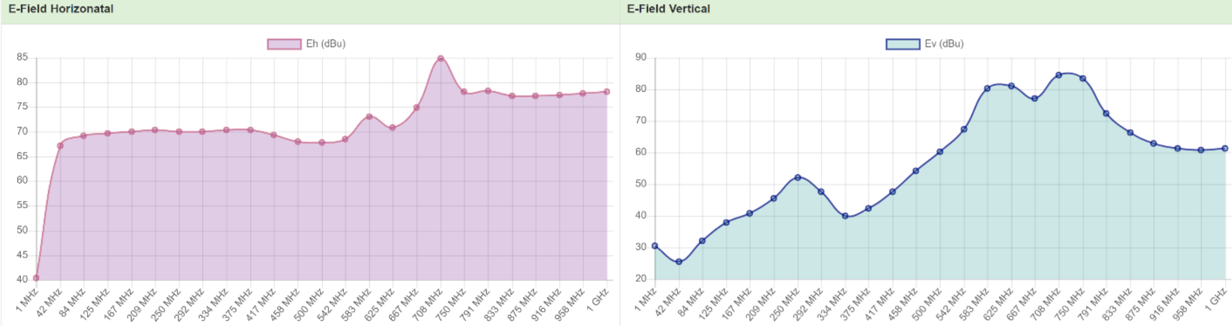


Figure 10-12: Executive summary of RE experiments

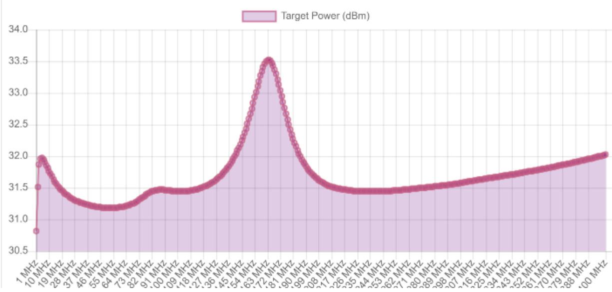
For RE experiments, the executive summary details the Eh and Ev profile.

10.4.3 Setup Details

This section details the setup of the experiment.

1. Test setup

BCI Method	Closed Loop
BCI mode	CommonModeBCI
Injection Clamp Location	900.000000mm
Monitor Clamp Location	50.000000mm
Target Monitor Clamp Current	100.000000mA



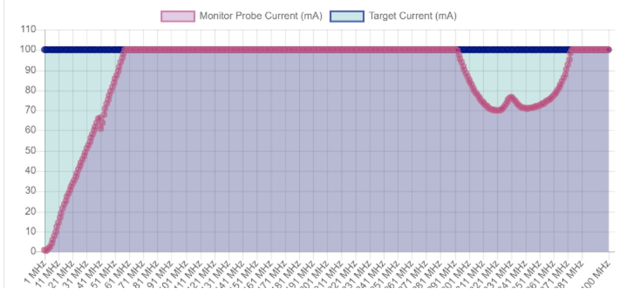
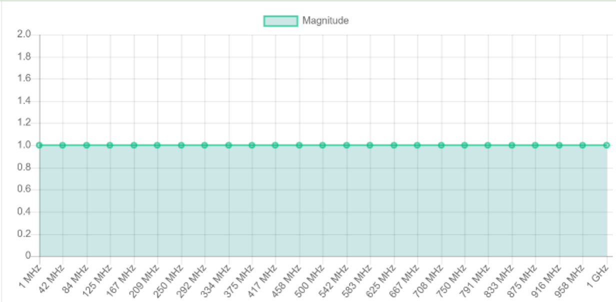


Figure 10-13: Test setup for BCI experiment

For RE experiment, this section details the excitation sources for the experiment and the RE standard property values.

1. Test setup

1.1 Source setup

Name	Frequency/Time	Pin 1	Pin 2	Signal
Source_0	FD	PCB.R8-1_0	PCB.R8-2_0	

1.2 Measurement setup

RE Standard	Cispr25re.json
Total Cable Length (mm)	100.000000
Straight Cable Length (mm)	100.000000
Cable Bend Angle (degrees)	90.000000
Distance of Cable from Table Edge (mm)	50.000000
DUT Height from Table (mm)	50.000000
Distance of Antenna from Cable Center (mm)	1000.000000
Height of Antenna from Ground (mm)	100.000000
Length of Table (mm)	200.000000
Height of Table (mm)	900.000000
Width of Table (mm)	100.000000

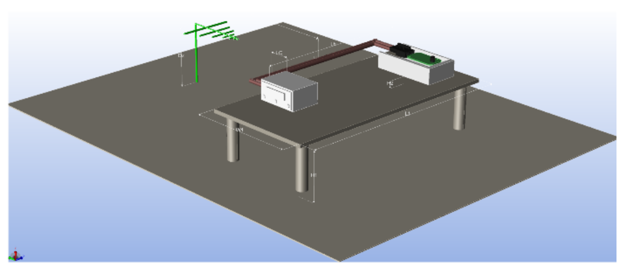


Figure 10-14: Test setup for RE experiment

10.4.4 Detailed Analysis

This section comprised of the following:

10.4.4.1 Component results

This section details the noise profile for all components.

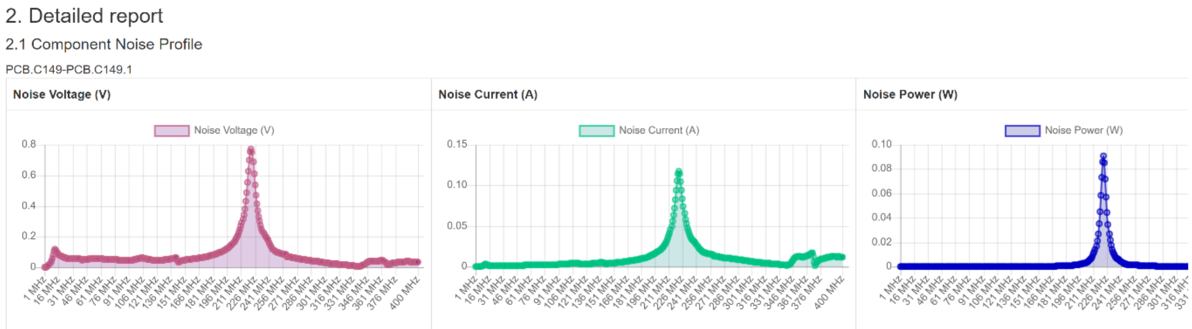


Figure 10-15: Detailed report – component noise

10.4.4.2 Cable noise (BCI only)

This section details the noise carried by the cables. This is specific to BCI experiment.

Figure 10-16: Detailed report – cable noise

10.5 RE results: Electric fields

For RE experiments, the simulated electric fields can be visualized from the "Results" section in the Project Tree Widget. Ex, Ey, Ez represents the x, y and z components of the radiated electric field at the antenna. Eh, Ev represents the horizontal and vertical components of the electric field, while Et represents the total electric field.

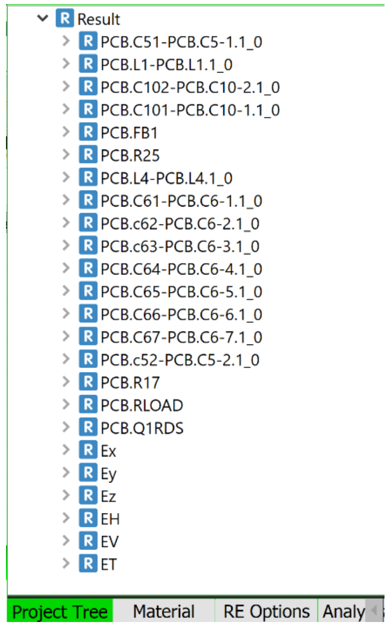


Figure 10-17: Electric field results

To visualize the field as a function of frequency, click on the arrow to the left of the desired field and double click on “E”.

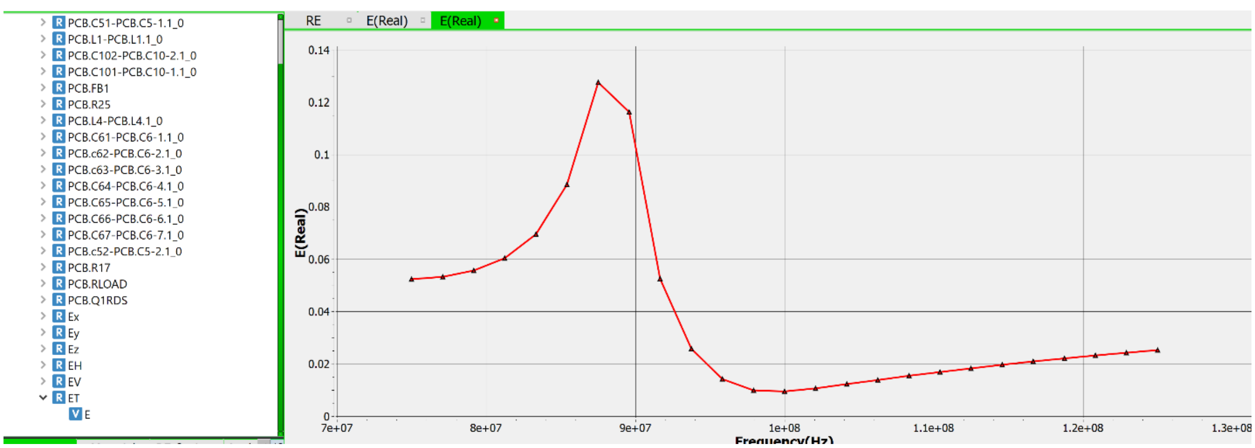


Figure 10-18: Electric field plots

In order to switch between real/imaginary/absolute/phase/dbu values, please go to plot options and select accordingly.

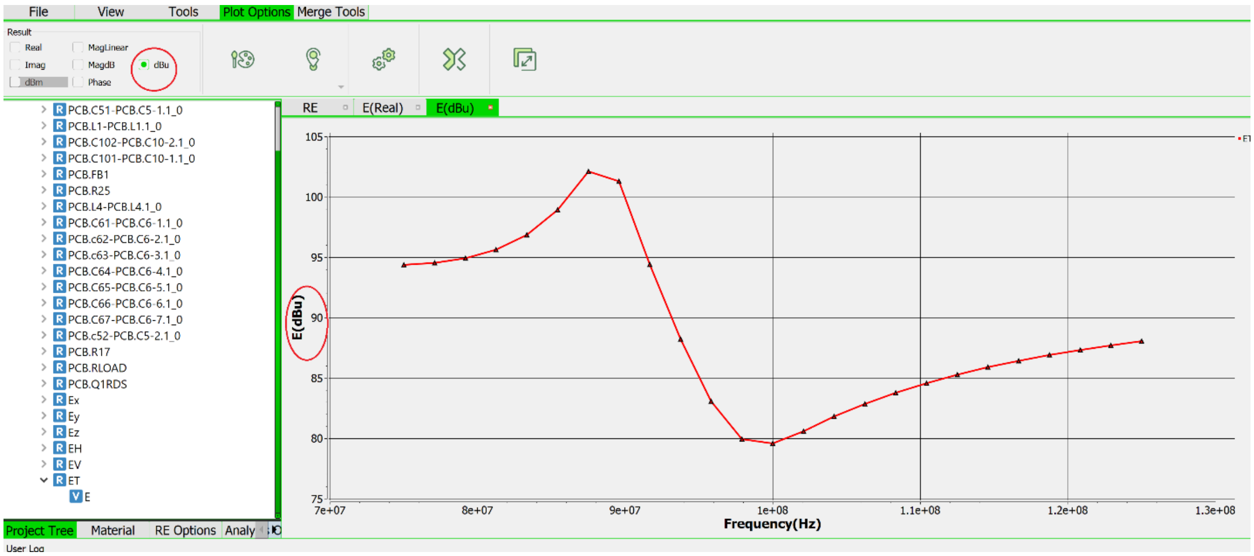


Figure 10-19: Different plot options for electric field

Limit-lines specific to the RE options (e.g. CISPR 25) are available in the project tree window.

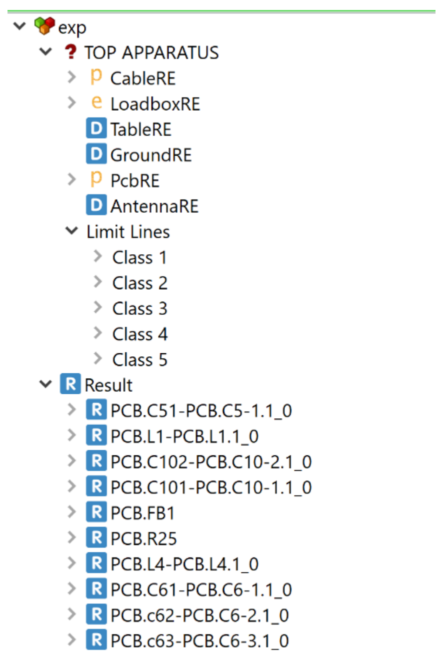


Figure 10-20: Limit lines available in the project tree window

Any of these lines can be plotted together with the simulated results to check Pass/Fail. Select the desired limit line from the list and drag it on top of the existing plot. Please note that the simulated fields need to be in dbu in order to plot with the limit lines.

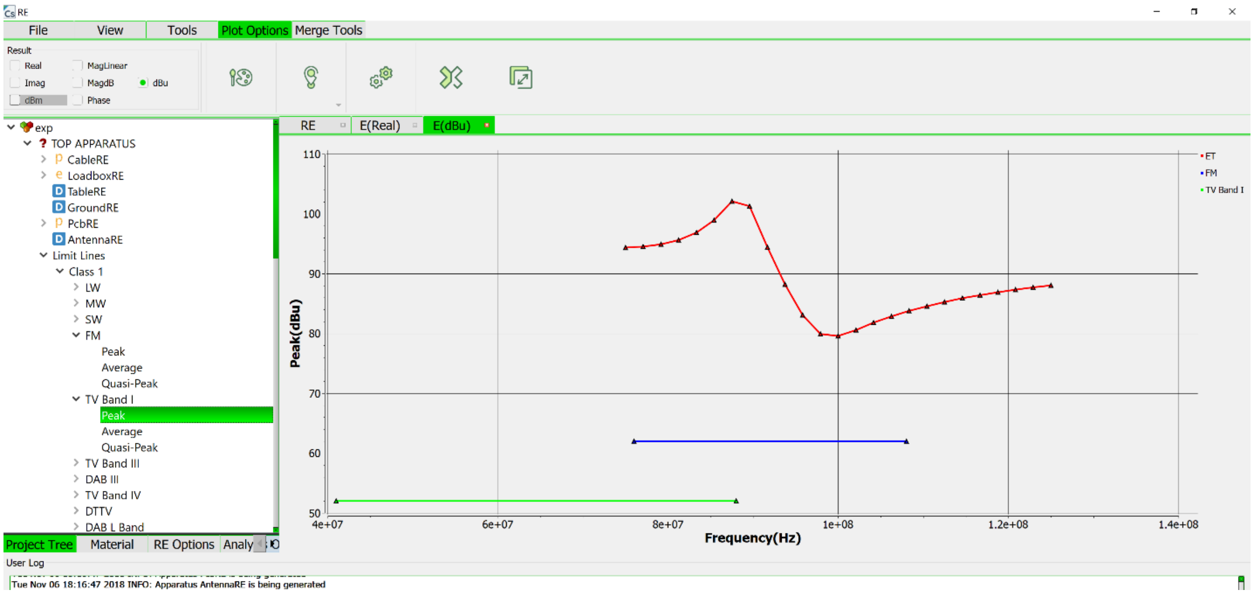


Figure 10-21: Plotting limit lines with the simulated electric field

11 Chapter 11: Example CompScope Experiments

11.1 Example 1: Full flow example with Simple Component Model

Movie: http://simyog.com/resources/examples/Example1/Example1.mp4

Resources: http://simyog.com/resources/examples/Example1/Example1_Resources.zip

11.2 Example 2: Full flow example with SPICE/Touchstone Component Model

Movie: http://simyog.com/resources/examples/Example2/Example2.mp4

Resources: http://simyog.com/resources/examples/Example2/Example2_Resources.zip

11.3 Example 3: Full flow example with ICIM Component

Movie: http://simyog.com/resources/examples/Example3/Example3.mp4

Resources: http://simyog.com/resources/examples/Example3/Example3_Resources.zip

11.4 Example 4: Full flow example with IBIS Component

Movie: http://simyog.com/resources/examples/Example4/Example4.mp4

Resources: http://simyog.com/resources/examples/Example4/Example4_Resources.zip

11.5 Example 5: Connector, Housing and PCB Merge

This example demonstrates the process of merging Connector with PCB and subsequently merging the Merged Apparatus (Connector + PCB) with Housing Apparatus.

Step 1: Import the PCB with the required nets, assign materials, add pins required for merge and finalize (0).

Step 2: Import the connector file, assign materials, add required pins and finalize (0).

Step 3: Import the housing file, assign materials, add required pins and finalize (6.7).

Step 4: Merge Connector Apparatus with PCB apparatus as explained in 8.2.

Step 5: Merge the Merged Apparatus (Connector + PCB) with Housing apparatus on similar lines as explained in **Error! Reference source not found.**.

The video below demonstrates the method explained above.

Movie: http://simyog.com/resources/examples/Example5/Example5.mp4

Resources: http://simyog.com/resources/examples/Example5/Example5_Resources.zip

12 Interpretation of Log Messages

Different types of ERROR messages. Operation user trying to perform is aborted.

- `ILLEGAL ENTRIES`
  - Operation halted because of either invalid user input or invalid internal data.
- `NOT IMPLEMENTED`
  - Unsupported feature exercised. Cannot proceed unless user avoids the error causing operation.
- `INVALID FUNCTION CALL`
  - Operation halted because of either invalid internal function call.
- `OUT OF MEMORY`
  - System memory limit reached. Fatal error as given system is incapable of solving user problem.
- `FILE NOT OPEN`
  - Inaccessible file or wrong file path given by user.
- `ERROR OPENING FILE`
  - CompScope is not having permission to access the file or file may not exists.
- `DIRECTORY ERROR`
  - Invalid directory path given by user.
- `LICENSE NOT FOUND OR EXPIRED`
  - Invalid license file. Fatal error as CompScope cannot be used unless this error is resolved.
- `UNDEFINED`
  - Unknown source of error. If encountered during solve it is advised that fresh experiment should be started.
- `ABORTED BY USER`
  - If user aborts ongoing process above error is displayed once the process stop. It might take few tens of seconds before process gets killed.

13 Appendix

13.1 BCI: IEC-62433-4 Standard

CompScope supports IEC-62433-4 IC Immunity Model (ICIM) for Conducted Immunity (CI). The details of the model are given in the detailed documentation of IEC-62433-4. The complete article can be found here: (https://webstore.iec.ch/publication/24943).

The abstract of the article available in open literature is provided below:

"IEC 62433-4:2016 specifies a flow for deriving a macro-model to allow the simulation of the conducted immunity levels of an integrated circuit (IC). This model is commonly called Integrated Circuit Immunity Model - Conducted Immunity, ICIM-CI. It is intended to be used for predicting the levels of immunity to conducted RF disturbances applied on IC pins. In order to evaluate the immunity threshold of an electronic device, this macro-model will be inserted in an electrical circuit simulation tool. This macro-model can be used to model both analogue and digital ICs (input/output, digital core and supply). This macro-model does not take into account the non-linear effects of the IC. The added value of ICIM-CI is that it could also be used for immunity prediction at board and system level through simulations. This part of IEC 62433 has two main parts:

- the electrical description of ICIM-CI macro-model elements;

- a universal data exchange format called CIML based on XML. This format allows ICIM-CI to be encoded in a more useable and generic form for immunity simulation."

In this document, the basic principles of IEC-62433-4 is discussed. The reader is encouraged to go through the full documentation for detailed understanding. For our discussion, let us consider an IC with 10 pins as shown in Figure 13-1.

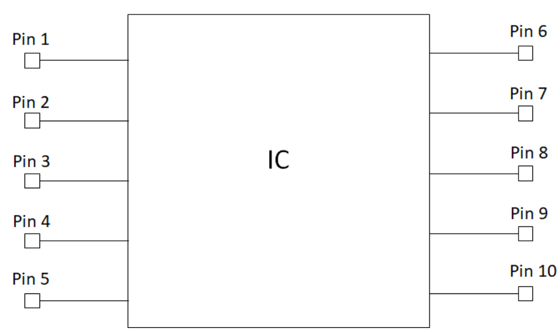


Figure 13-1: IC with 10 pins for demonstration

The basic purpose of IEC-62433-4 is to provide a modeling-framework in which a typical IC can be modeled for a BCI simulation. A complete black-box representation of the IC is difficult to generate:

(a) Why not a complete S-parameter block: The IC contains non-linear elements, it cannot be represented by a Linear, Time Invariant (LTI) black-box. In other words, for the example IC of Figure 13-1, a 10 × 10 S-parameter block cannot be used to macro-model the IC.

(b) Why not a complete SPICE sub-ckt: This is possible in theory. However, the IC manufacturer will not be willing to share the detailed information of the internals of the IC. Also, most ICs contain more than million transistors along with parasitic elements which will be extremely time consuming to process in a BCI simulation.

Therefore, IEC-62433-4 resorts to modeling an IC for BCI simulation using 2 types of models:

13.1.1 ICIM-CI PDN model

The "Passive Distribution Network" (PDN) represents the impedance seen at any "Disturbed Input" (DI) of the corresponding IC.

The PDN can be used to define the impedance of:

(a) 1 pin with respect to its ground, as shown in Figure 13-2, for Pin 1 (DI 1) and its ground Pin 5 (GND for DI 1). The PDN itself can be represented by either (i) a SPICE subckt (ii) a touchstone file (here s1p file).

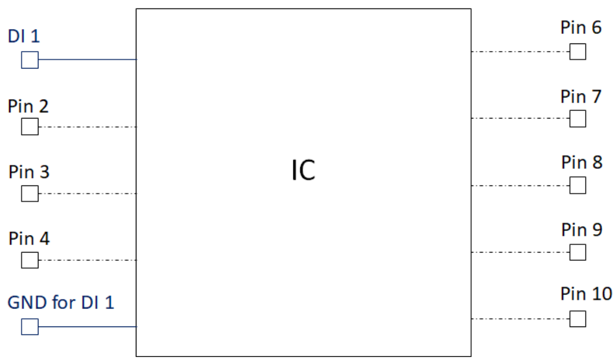


Figure 13-2: 1 pin DI model

(b) 2 differential pins with respect to their ground, as shown in Figure 13-3, for Pins 2 and 3 and its ground Pin 5. The PDN itself can be represented by either (i) a SPICE subckt (ii) a touchstone file (here s2p file).

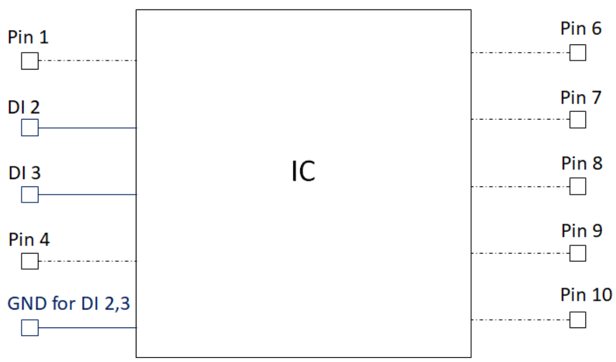


Figure 13-3: 2 pin DI model

(c) multi-pin model with respect to their ground.

This modeling methodology enables the computation of impedance as seen by the disturbed IC pins without the need for obtaining an entire transfer function of the IC itself. The PDN values themselves can be computed by VNA measurements on the IC pins or using circuit simulation tools if the circuit level description of the IC is available.

13.1.2 ICIM-CI IB model

The “Internal Behavior” (IB) model specifies the pass/fail limit lines for the specific IC. Let us denote the IB for aggressor pin “i” and victim pin “j” as IB (j, i). Then, the IB (j, i) represents the “Power vs Frequency” profile which when applied to Pin i, will cause a failure in Pin j. This, therefore represents a “limit-line” for the disturbed input (Pin i).

The IB model can be generated by using DPI tests on the IC or from circuit simulation if the circuit details of the IC is available.

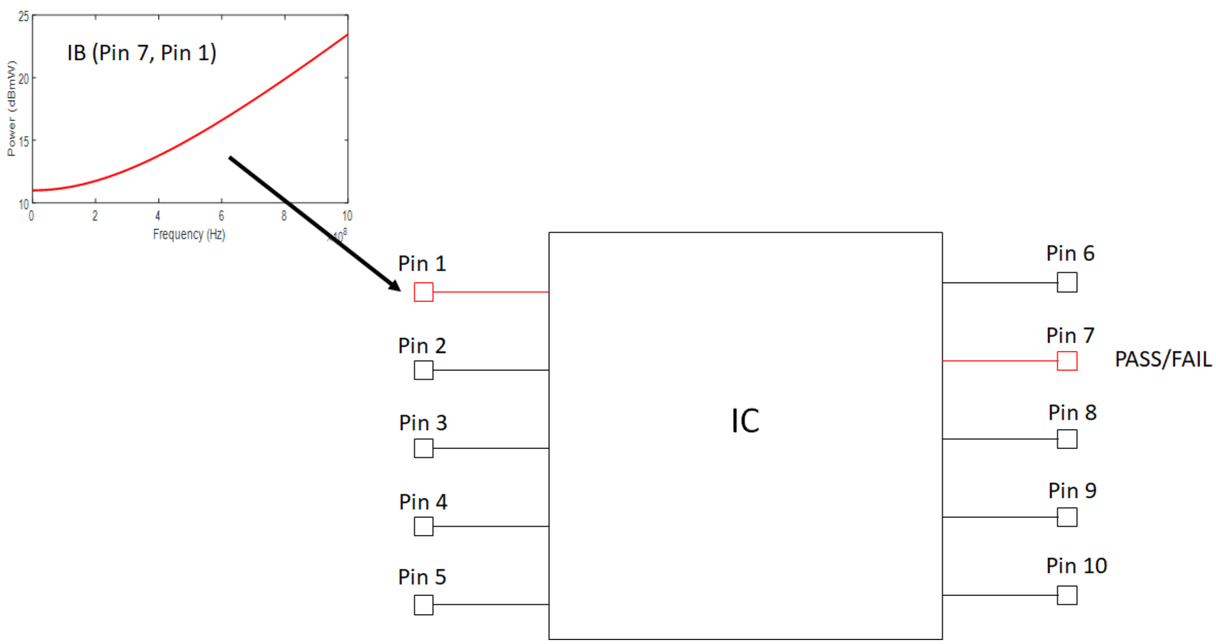


Figure 13-4: Example IB profile

In the above example, shown in Figure 13-4, the profile of power in dBmW is generated for aggressor pin Pin 1 for the victim Pin 7. If the noise power level at IC Pin 1 exceeds that in the IB, it will generate a failure in Pin 7. The pass-fail region corresponding to this IB is shown in Figure 13-5.

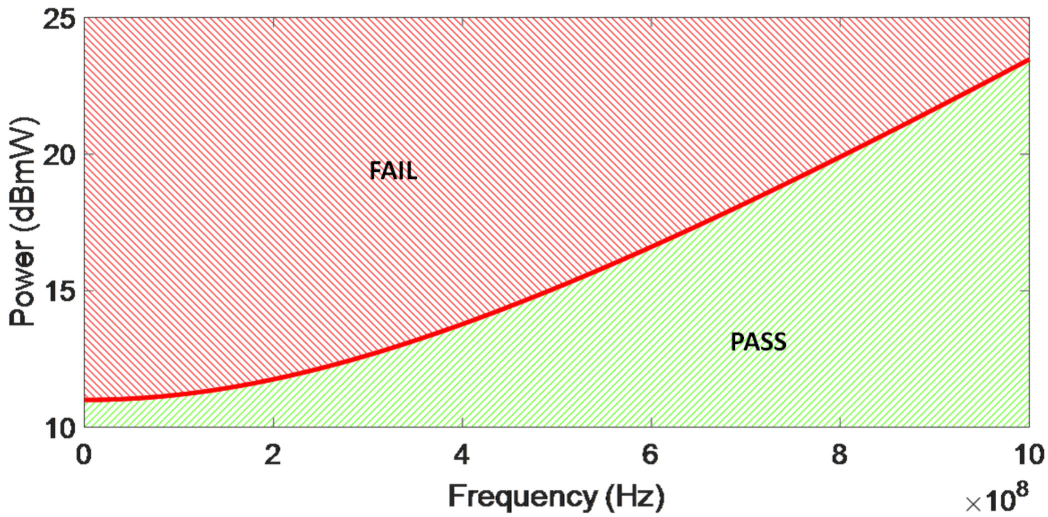


Figure 13-5: Pass fail regions for an IB

13.2 RE: IEC-62433-2 Standard

13.3 BCI: F-140 Clamp

13.3.1 Explanation of CompScope injection clamp library model

CompScope injection clamp library uses equivalent circuit/electromagnetic models for injection clamps generated from measurements performed on the injection clamp. For each clamp, CompScope uses 2 different network-parameter models:

13.3.1.1 Injection clamp to cable coupling model

The injection clamp to cable model is captured through measurement as an s3p file. This data is used to generate applicable snp file for any multi-wire harness case using proprietary technology.

To generate the equivalent model, measurement is performed with a single small cable kept inside the injection clamp as shown in Fig. 1. Three port measurement is performed using a network analyzer. The portion of the cable jutting out of the injection clamp and the L-clamps are de-embedded to generate the library s3p model for the particular injection probe.

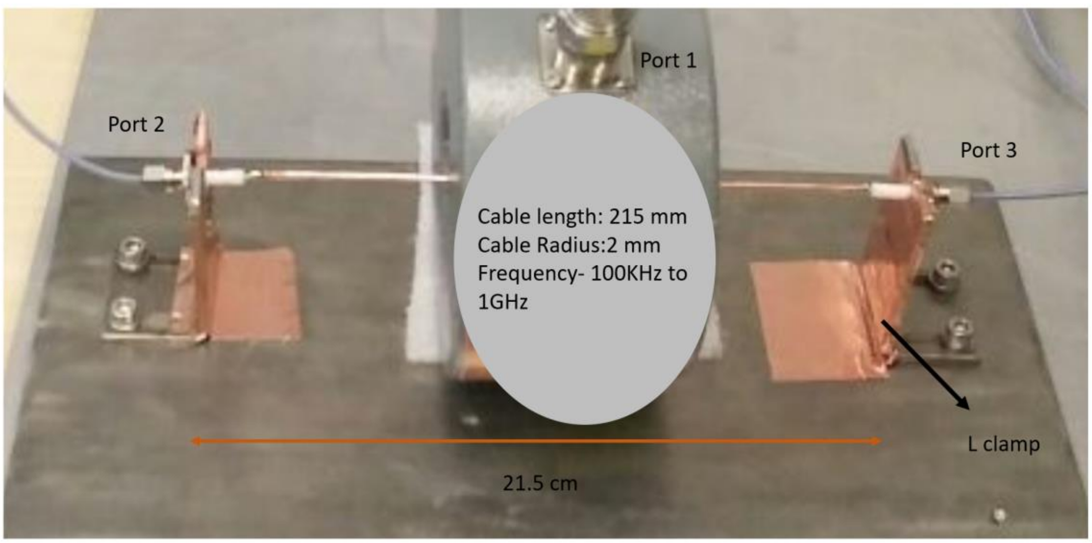


Fig. 1. S parameter measurement with cable inside clamp

Representative de-embedded measurement data for F140 clamp and a single cable with radius 2mm is shown below:

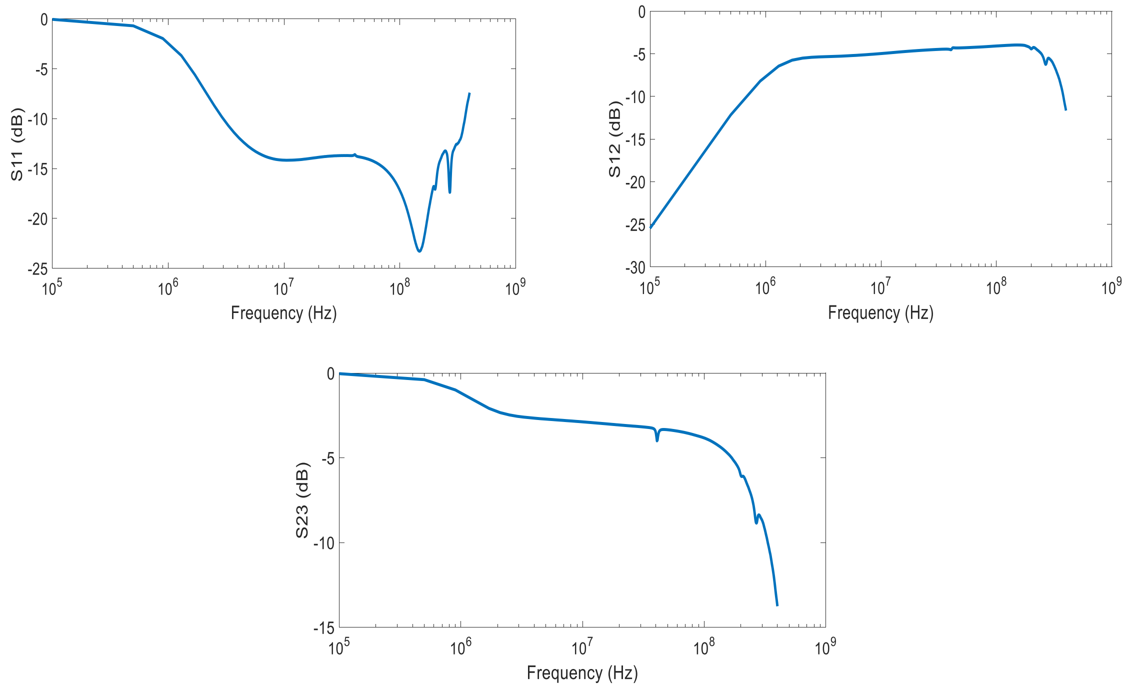


Fig. 2: Measured 3-port S-parameter data for F140

13.3.1.2 Injection clamp calibration model

The injection clamp calibration model is used for calibration to determine power level in open loop and closed loop simulation.

To obtain this calibration model, S-parameter measurement is done with bulk current injection probe fixture as shown in Fig. 3.

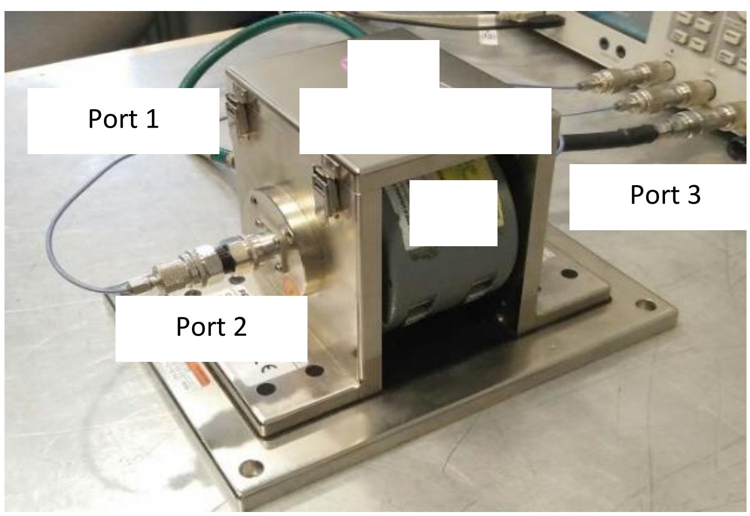


Fig. 3. S parameter measurement with fixture

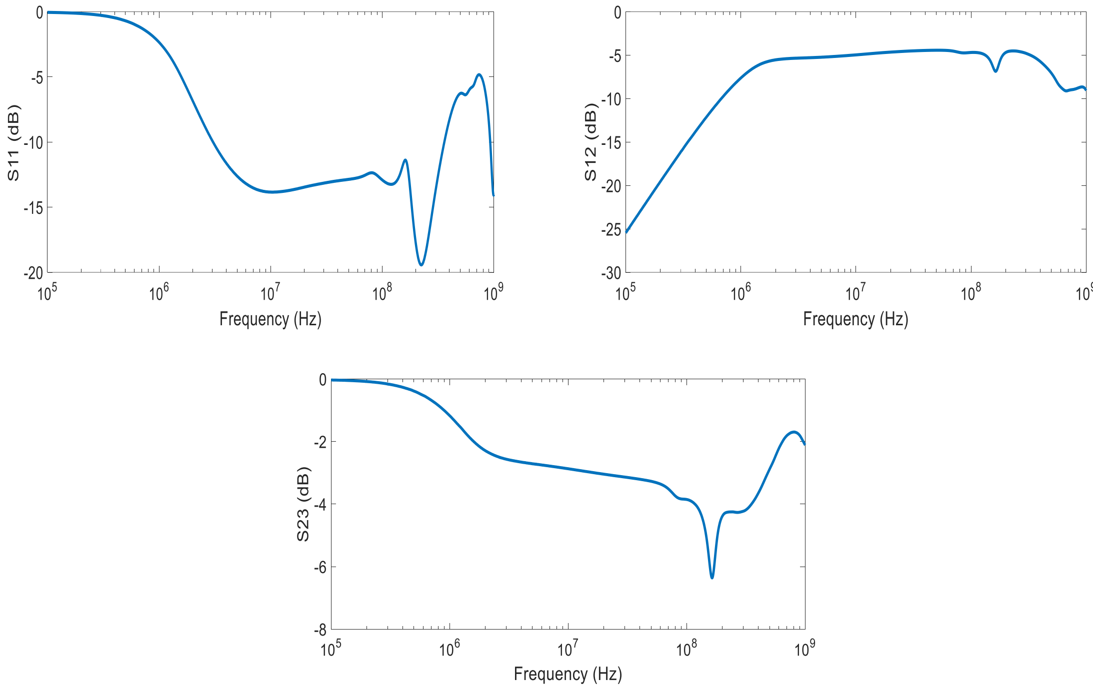
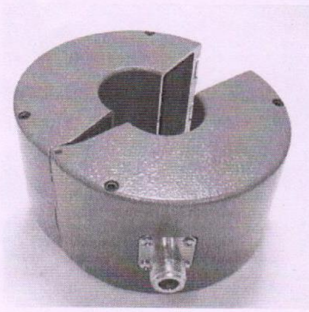


Fig. 4: Measured 3-port S-parameter data for F140 with fixture (observe the match of S12 with datasheet Fig 5 insertion loss)

13.3.1.3 Injection clamp datasheet

This data is available from the manufacturer and is optional as long as the injection clamp dimensions as set in Table 1 is known.

Bulk Current Injection Probe



The F-140 is used for BCI testing in accordance with automotive specifications like SAE J15447 as well as diagnostic bulk current testing up to 1.3 GHz. The useable frequency range is 10 kHz to 1.3 GHz with a typical insertion loss of 7 dB or less from 700 kHz to 500 MHz. The CW input power rating is 100 watts for a duration of 30 minutes.

F-140 Specifications

Frequency:	100 kHz – 1 GHz
Inner Diameter:	40mm
Outer diameter:	127mm
Height:	70mm
Input Power rating:	100 watts

Accessories

Calibration Fixture: FCC-BCICF-2
Bandwidth: 100 kHz – 1 GHz

Current Probe: F-61
Bandwidth: 1 MHz – 1 GHz

F-140

Fischer Custom Communications, Inc.

Fig 5. Example datasheet (F140)

13.3.2 Data required for building a clamp model into CompScope library

Any one of the following options may be used for generating data for building CompScope injection clamp library model.

Option 1:

(1) 3D geometry (sat file) for the desired clamp

Option 2:

(1) De-embedded s3p file for injection clamp-single cable

(2) Injection clamp calibration s3p file

+

If available the datasheet of the clamp (similar to Fig. 5)

Table 1: Data required for building CompScope clamp model

Clamp	**Dimension**
	Outer Diameter of clamp
	Inner Diameter of clamp
	Clamp Length
Cable	Cable length (if not de-embedded)
	Cable radius
	Height from measurement plate
L-clamp	Height (if not de-embedded)
	Width (if not de-embedded)
Measurement (100KHz to 1GHz)	**Touchstone (s3p)**
Clamp calibration with fixture	S3p file
Clamp to cable coupling	S3p file