SPEED2000 EMI Simulation Tutorial

Product Version 16.6
July 2013

Document Updated on: June 6, 2013

Table of Contents

1 Introduction

This tutorial demonstrates how to use Electromagnetic Interference (EMI) mode in SPEED2000 to analyze the conducted and radiated noise from a DIMM memory module. It mainly introduces the setup of IBIS driver models for controller and memory.

1.1 Overview

The purpose of **EMI** workflow is to allow user to study the radiation from their designs. It lets user to study:

- Radiation versus frequency from the design
- Near-field E/H densities

The **EMI** workflow in SPEED2000 leads you to:

- Setup layout
- Setup simulation parameters
- Check errors
- Run simulation
- View results

1.2 Sample Case

The following three original files are used in this tutorial:

- SODIMM_EMI.spd – layout file for SODIMM
- It is located in: <INSTALL_DIR>\SpeedXP\Samples\SPEED2000\EMI Simulation\Examples_PreSetup\
- dram.ibs – IBIS file containing DRAM buffer models
- ctrl.ibs – IBIS file containing Ctrl buffer models
- They are located in: <INSTALL_DIR>\SpeedXP\Samples\SPEED2000\EMI Simulation\Examples_PreSetup\IBIS\

And the completed sample and IBIS files (with step by step setup introduced in this tutorial) are also provided and located in:

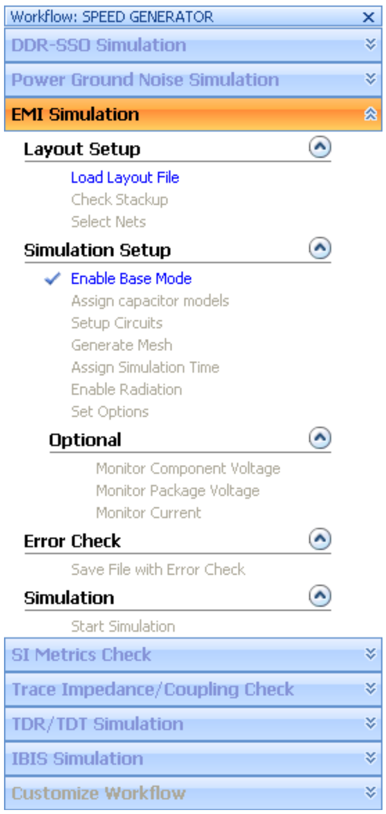
- <INSTALL_DIR>\SpeedXP\Samples\SPEED2000\EMI Simulation\Examples_PostSetup\

2 Layout Setup

This chapter describes how to set up layout for EMI simulation.

2.1 Loading Layout File

1. Launch **SPEED2000 Generator**.
2. Select the **EMI Simulation** workflow.



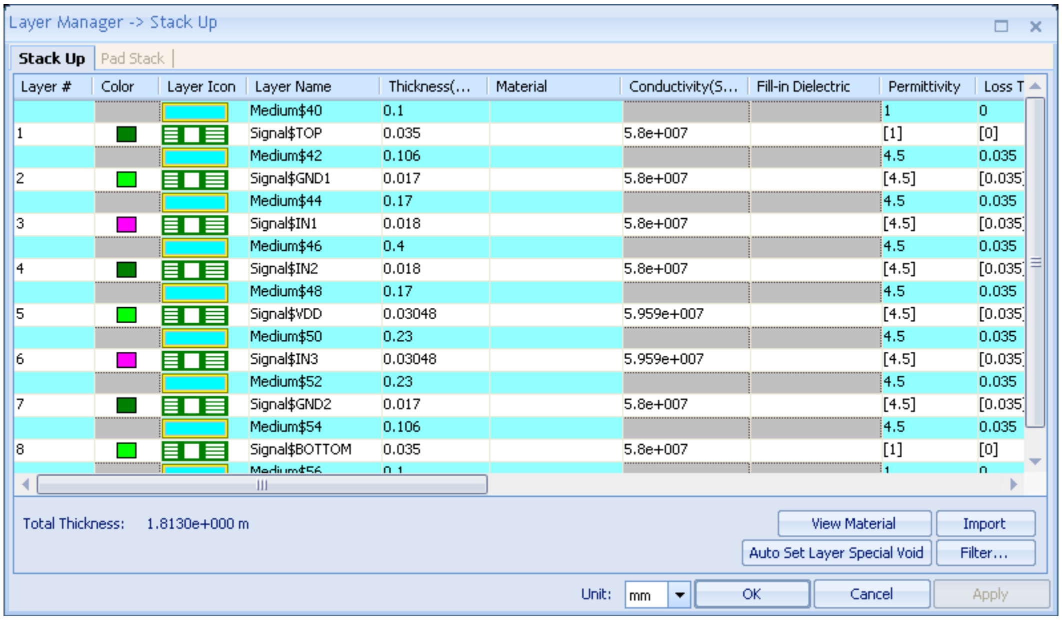
3. Click **Load Layout File** to load .spd file.

2.2 Checking Stackup

This section leads you to set up the stackup and the parameters, which should be set correctly for each simulation.

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

   The **Layer Manager -> Stack Up** window opens.

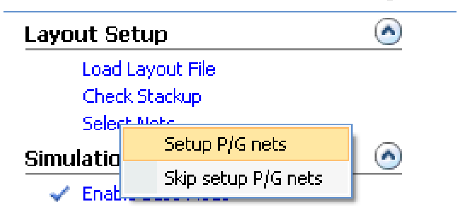


2. Check and edit the stackup as desired (no change in this example).
3. Click **OK** to exit the window.

2.3 Selecting Nets

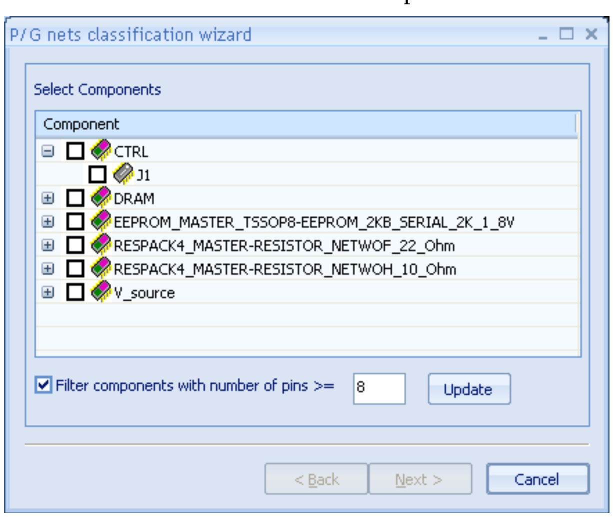
This section leads you to classify and select the nets to be simulated. The power and ground nets need to be properly assigned in their respective groups before performing simulation.

1. Click **Select Nets** in the **Workflow** pane.



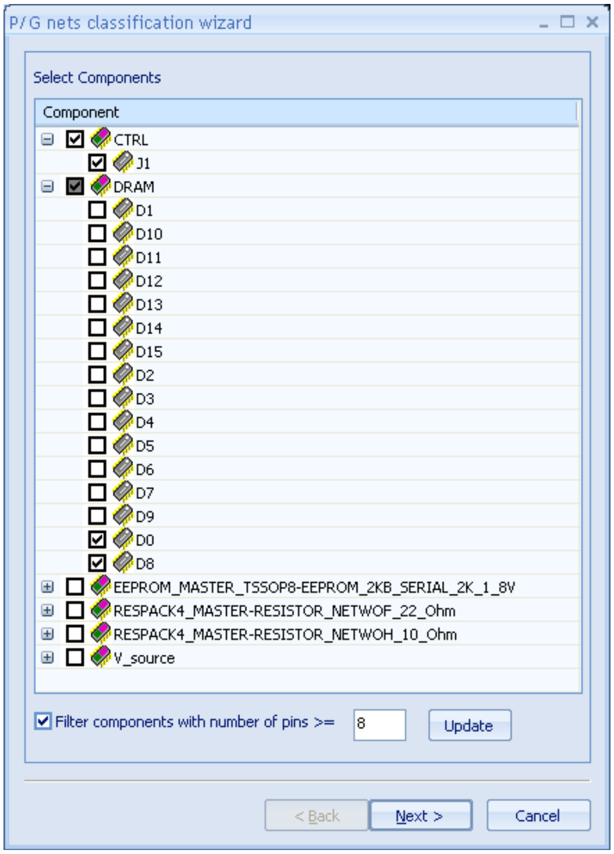
2. Click **Setup P/G nets** in the pop-up menu.

   The **P/G nets classification wizard** opens.



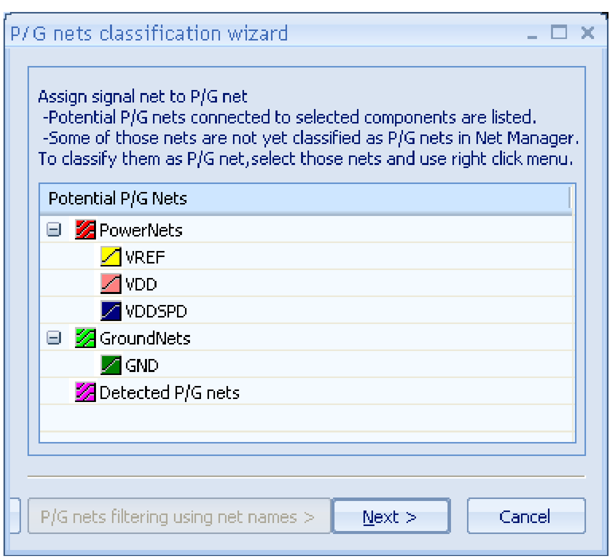
Tips	You can also open the **P/G nets classification wizard** by right-clicking a net in **Net Manager** and choosing Classify > P/G nets classification wizard….

3. Select **J1**, **D0,** and **D8**.



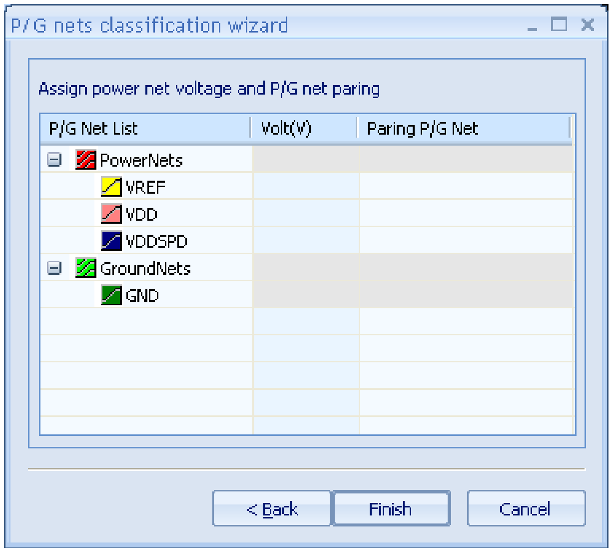
4. Click **Next**.

The nets connected to a given component on more than one connection or ball are detected as potential Power or Ground nets.



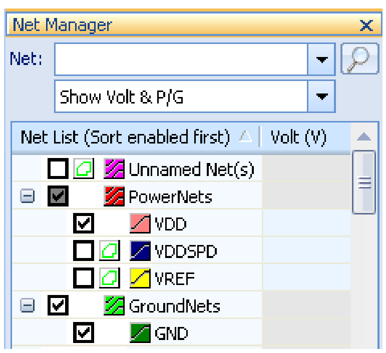
5. Click **Next**.

All the power and ground nets defined and generated are shown.

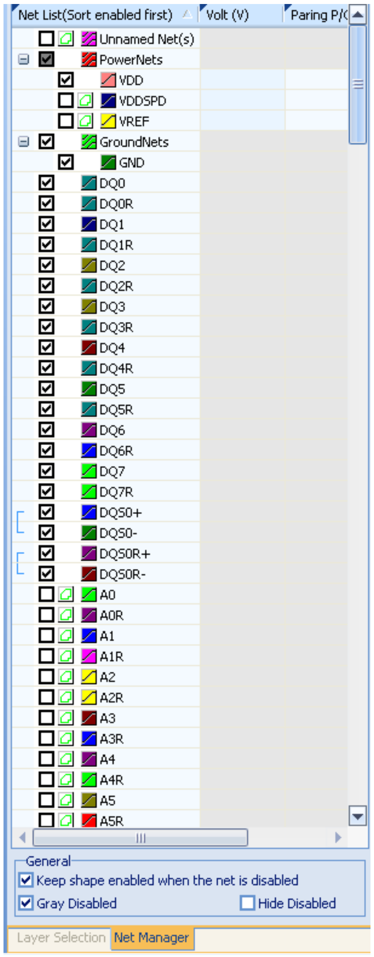


6. Click **Finish**.

The **Net Manager** automatically appears in the right pane.



7. Check the signal nets: **DQ0~DQ7**, **DQ0R~DQ7R**, **DQS0+**, **DQS0R+**, **DQS0-,** and **DQS0R-**.



All the **DQ** and **DQS** signal nets connected to **U0** and **U8** are enabled.

3 Simulation Setup

3.1 Enabling Base Mode

The EMI analysis relies on the base analysis mode. It is similar to other analysis performed in the base mode. The only difference is to enable the storage of radiation results and potentially add probes in locations that typically would not be probed in a signal integrity-only context.

Click **Enable Base Mode** in the **Workflow** pane before setting up other simulation options.

When enabled, a check mark ✓ appears next to the workflow step.

3.2 Assigning Capacitor Models

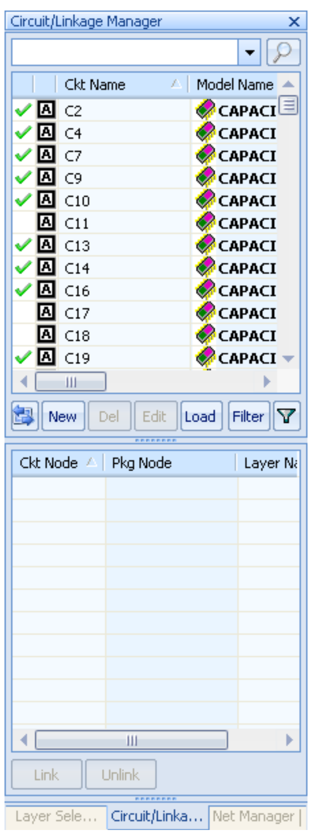
There are two ways to assign spice models to capacitors in SPEED2000:

- Entering the capacitor netlist on the fly
- Importing a library directly

This section introduces how to assign spice models to all relevant capacitors with the first method.

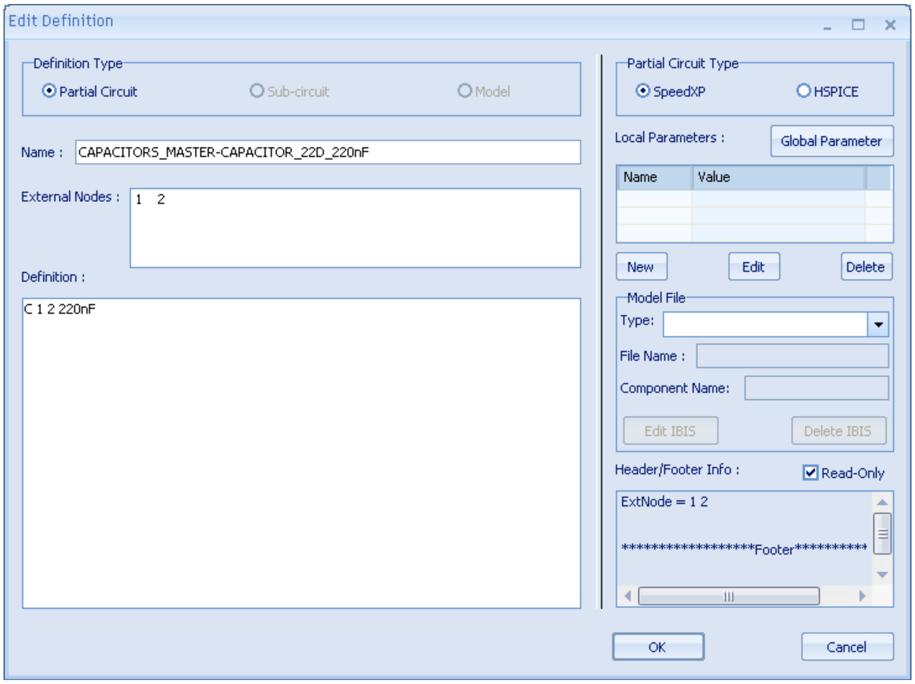
1. Choose Setup > Circuit/Linkage Manager….

   The **Circuit/Linkage Manager** opens.



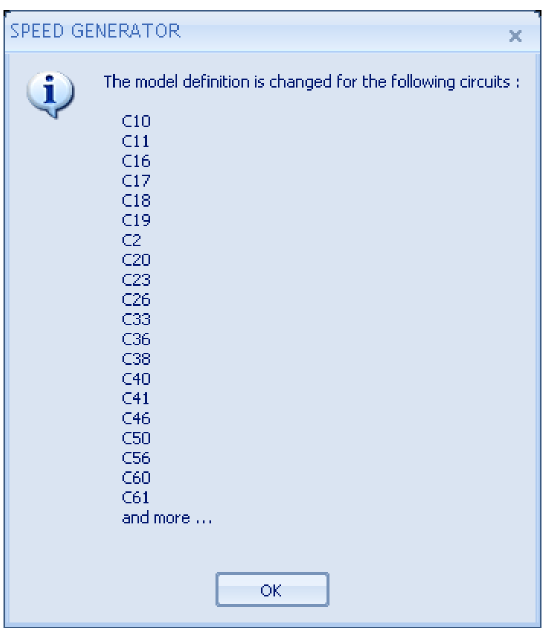
2. Select **C2**.
3. Click **Edit.**

   The **Edit Definition** dialog box appears.



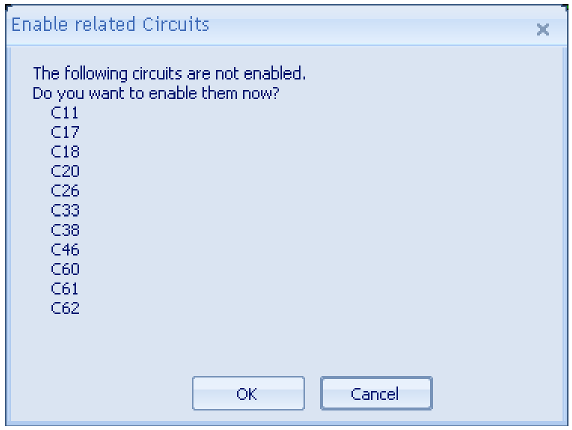
4. Define a simple ideal capacitor as:
   - **Name:** CAPACITORS_MASTER-CAPACITOR_22D_220nF
   - **External Nodes:** 1 2
   - **Definition:** C 1 2 220nF
5. Click **OK**.

   The **SPEED GENERATE** dialog box appears, showing all capacitors with the same part number or model name are updated to use the given model.



6. Click **OK**.

   A dialog box prompts to ask you whether to enable other related circuits.



7. Click **OK**.
8. Repeat Step **2** to **5** to define models for all the capacitors in this design.

NOTE!	So far, the ideal capacitor models are used for all capacitors. However, it is recommended to use parasitic to make the simulations more realistic.

3.3 Setting Up Circuits

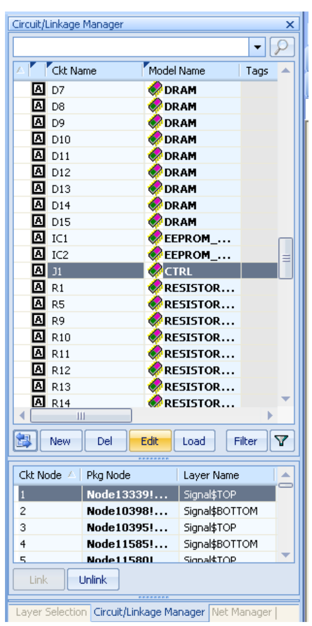
This section defines the controller and memory device models based on the IBIS models. The design is set up in a write mode, that is, the controller is writing to the memories running at DDR3-1333 speed.

3.3.1 Setting Up Controller Model

1 Choose Setup> Circuit/Linkage Manager….

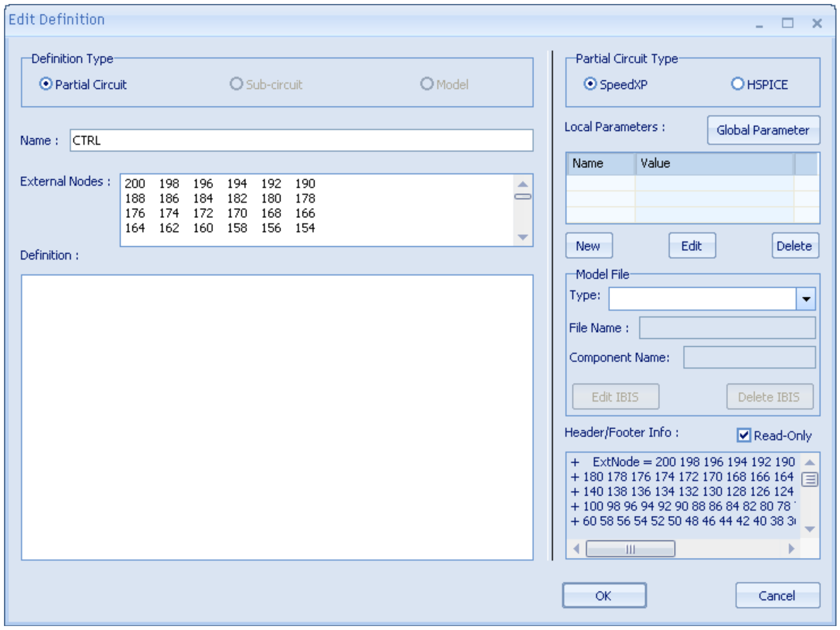
The **Circuit/Linkage Manager** window opens.

2 Select **J1.**

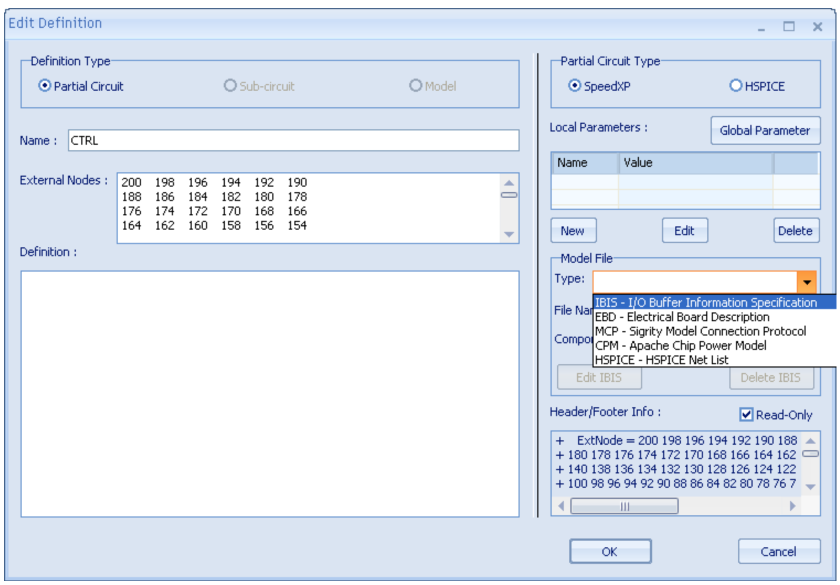


3 Click the **Edit** button.

The **Edit Definition** window opens.

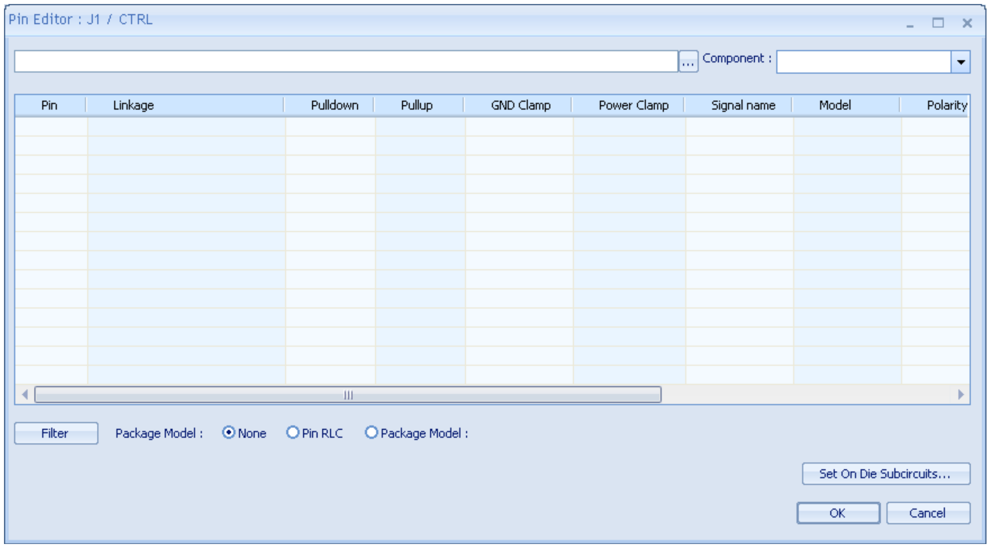


4 Select **IBIS- I/O Buffer Information Specification** from the **Model File Type** drop-down menu.



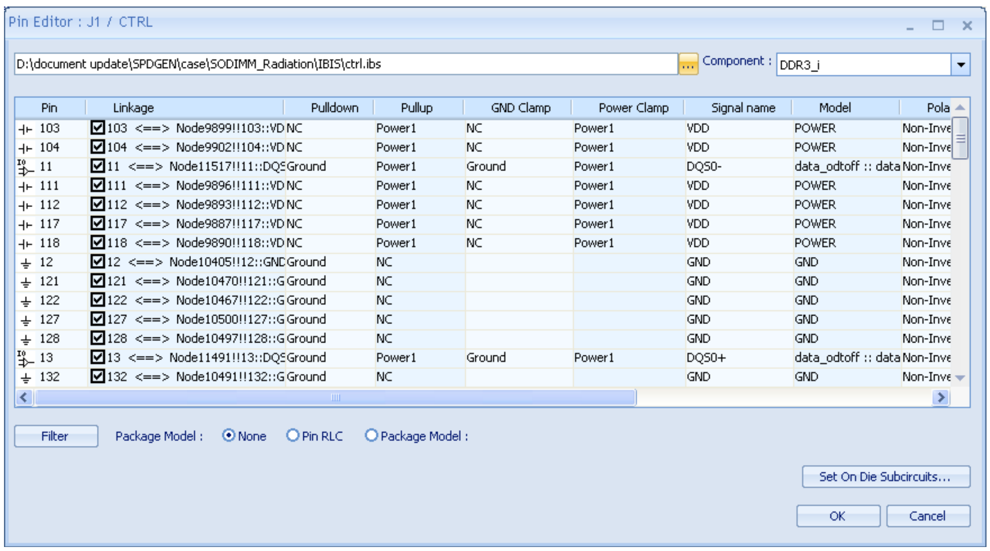
5 Click the **Edit IBIS** button.

The **Pin Editor: J1 / CTRL** window opens.



6 Click the browse button  and select the file ctrl.ibs for **J1**.

The **Pin Editor** field is updated with the IBIS component information.

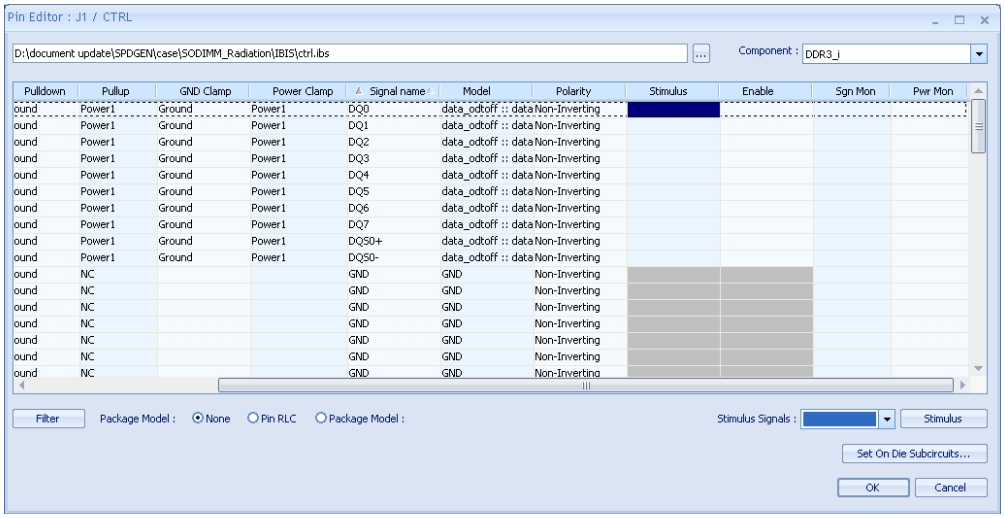


7 Define the stimulus for the controller **DQ** and **DQS** signals, assuming that all signals toggle in phase with a 1010.. pattern.

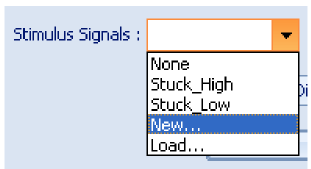
7.1 Click the **Signal Name** column to rank the signals, which makes it easier to identify **DQ** signals.

7.2 Click the blank cell in the **Stimulus** column.

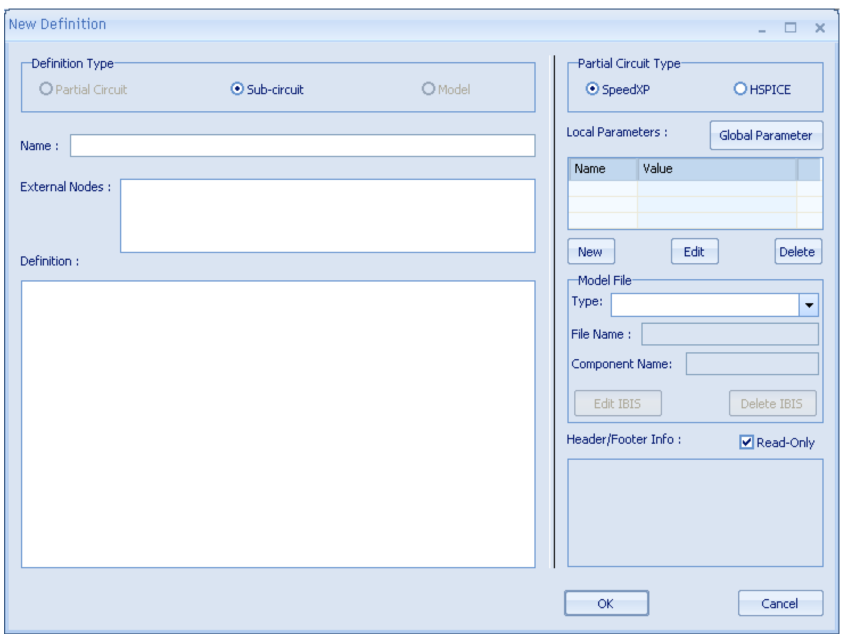
The **Stimulus Signals:** box appears in the lower right corner.



7.3 Select **New...** for **Stimulus Signals** from the drop-down menu.



The **New Definition** window opens.

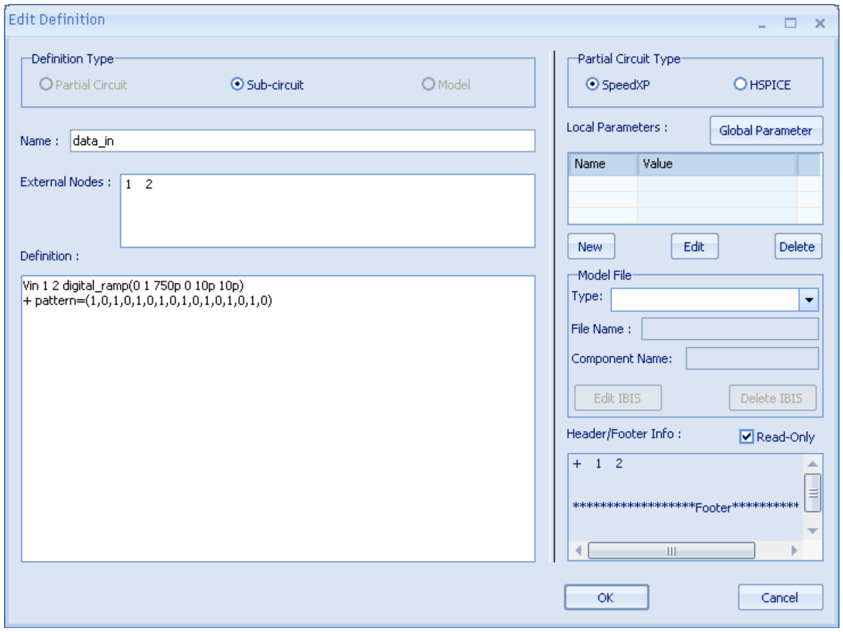


7.4 Input the following:

- **Name:** data_in
- **External Nodes:** 1 2
- **Definition:**

  Vin 1 2 digital_ramp(0 1 750p 0 10p 10p)

  +pattern=(1,0,1,0,1,0,1,0,1,0,1,0,1,0,1,0)



7.5 Click **OK**.

The new stimulus circuit **data_in** is created.

7.6 Assign signals **DQ0~DQ7** with **data_in**.

8 Repeat Step **7.2** to **7.6** to create new stimulus circuit with opposite polarity for **DQS+** with the following:

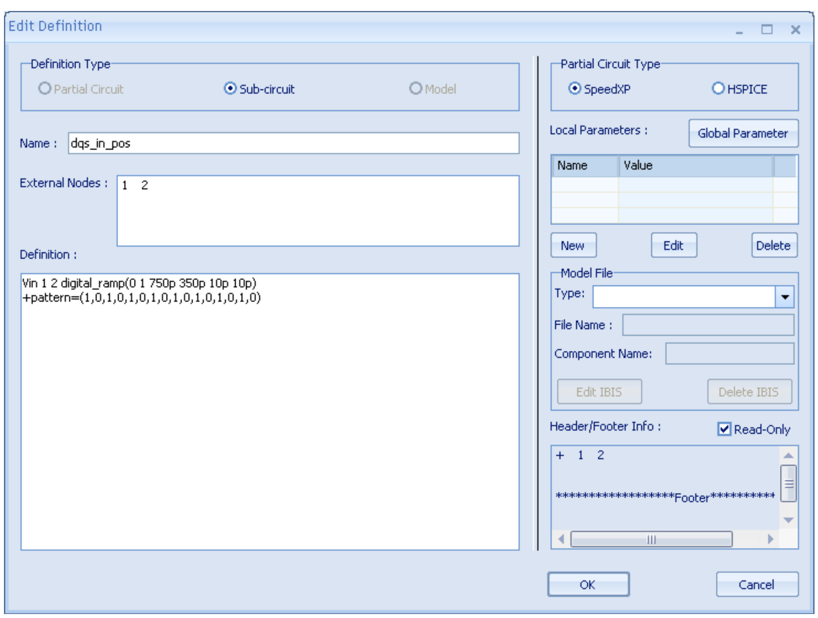
Name: dqs_in_pos

External Nodes:1 2

Definition:

Vin 1 2 digital_ramp(0 1 750p 375p 10p 10p)

+pattern=(1,0,1,0,1,0,1,0,1,0,1,0,1,0,1,0)



9 Repeat Step **7.2** to **7.6** to create new stimulus circuit with opposite polarity for DQS-: with the following:

Name: dqs_in_neg

External Nodes: 1 2

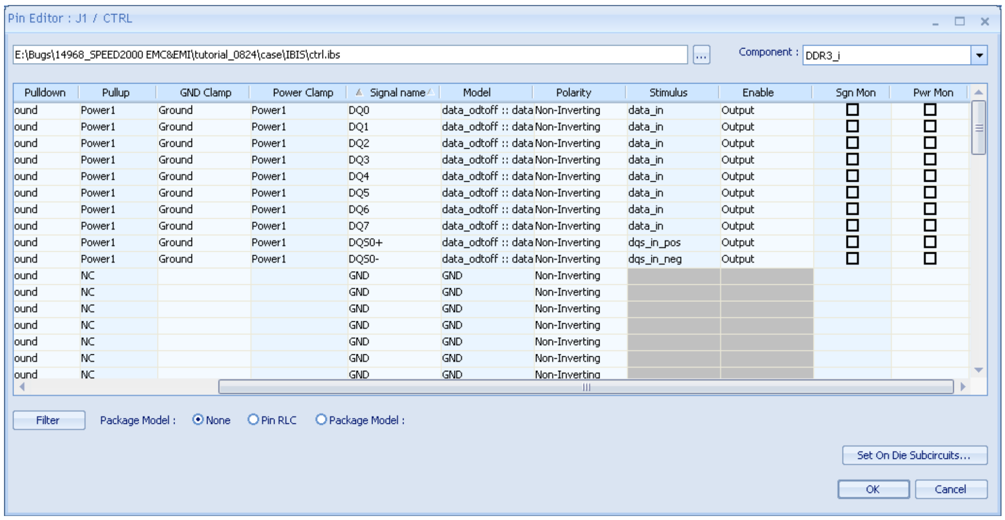
Definition:

Vin 1 2 digital_ramp(0 1 750p 375p 10p 10p)

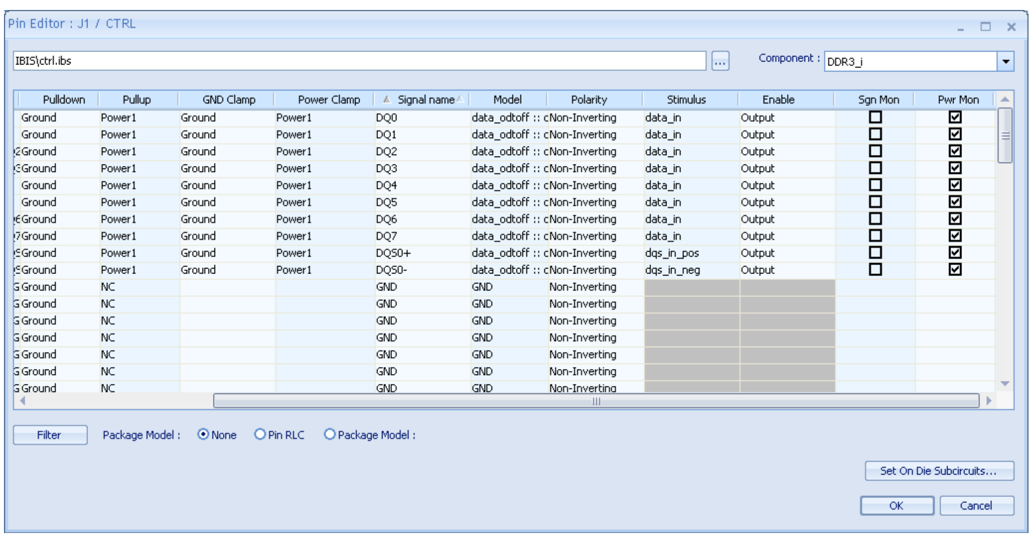
+pattern=(0,1,0,1,0,1,0,1,0,1,0,1,0,1,0,1)

10 Select **DQ0~DQ7**, **DQS0+**, and **DQS0-**.

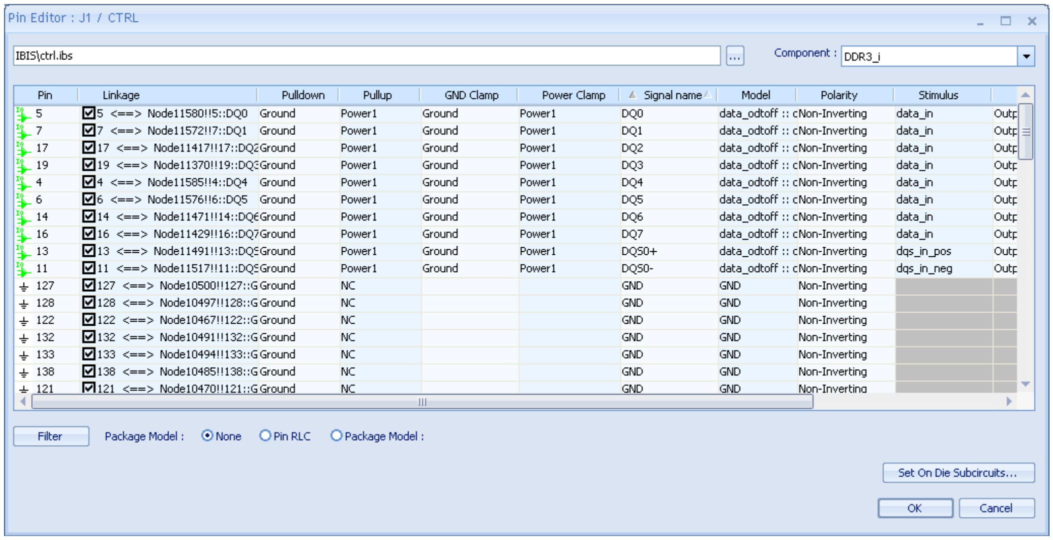
11 Select **data_odtoff** in the column of **Model**, and **Output** in the column of **Enable**.



12 Check **Pwr Mon**.

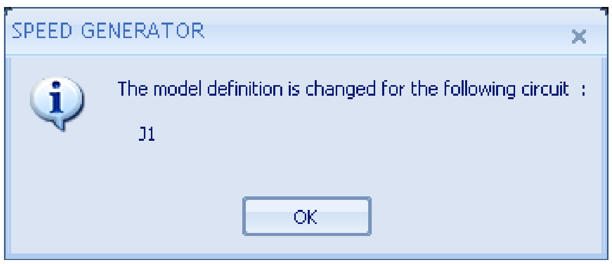


After all the settings, the green signs appear ahead of the selected signals.



13 Click **OK**.

A dialog box prompts to confirm that the model definition is changed.



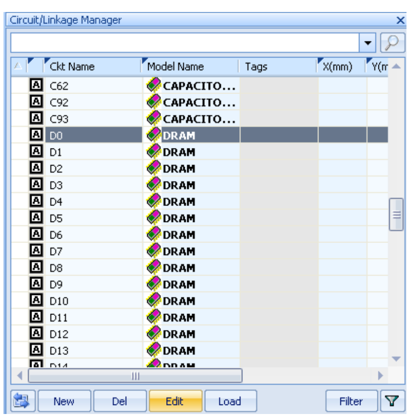
14 Click **OK**.

3.3.2 Setting Up Receiver Model

1. Choose Setup > Circuit/Linkage Manager….

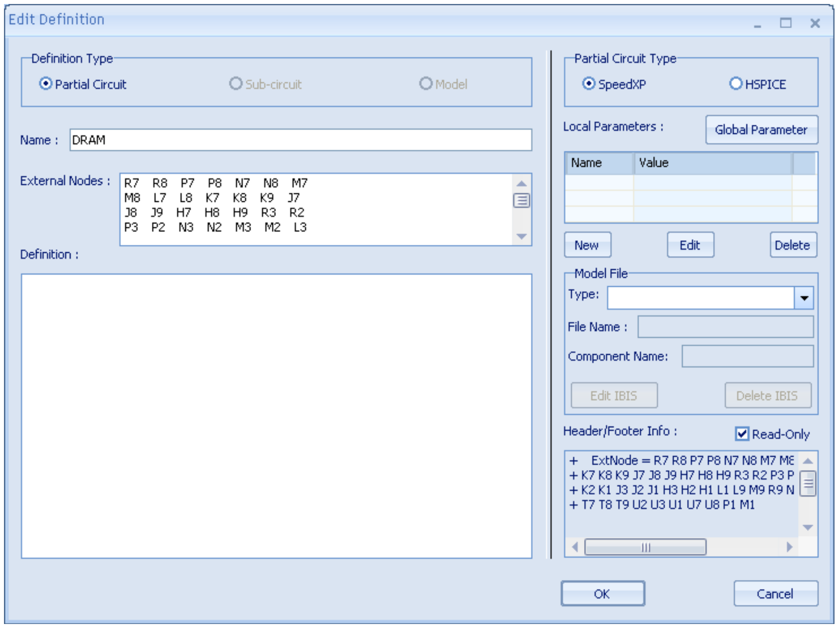
   The **Circuit/Linkage Manager** window opens.

2. Select **D0**.

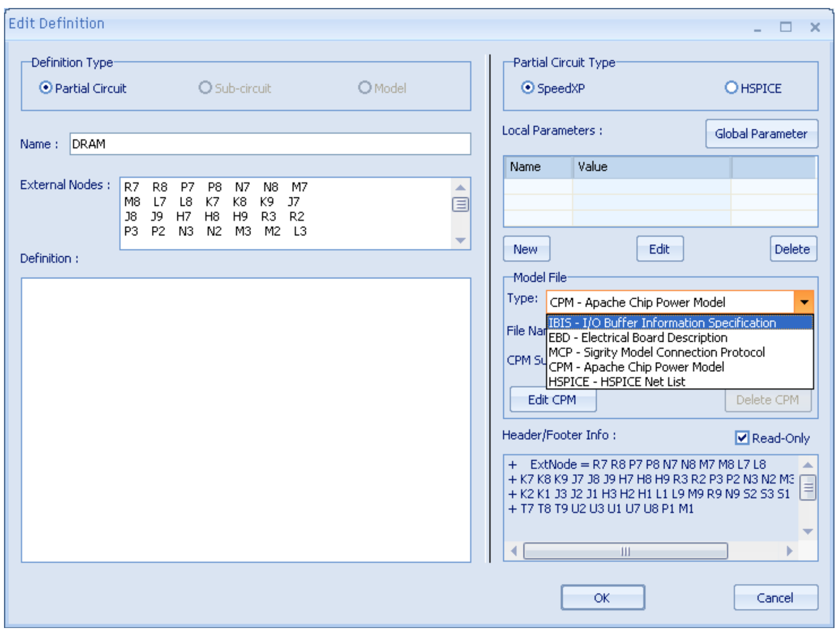


3. Click the **Edit** button.

   The **Edit Definition** window opens.

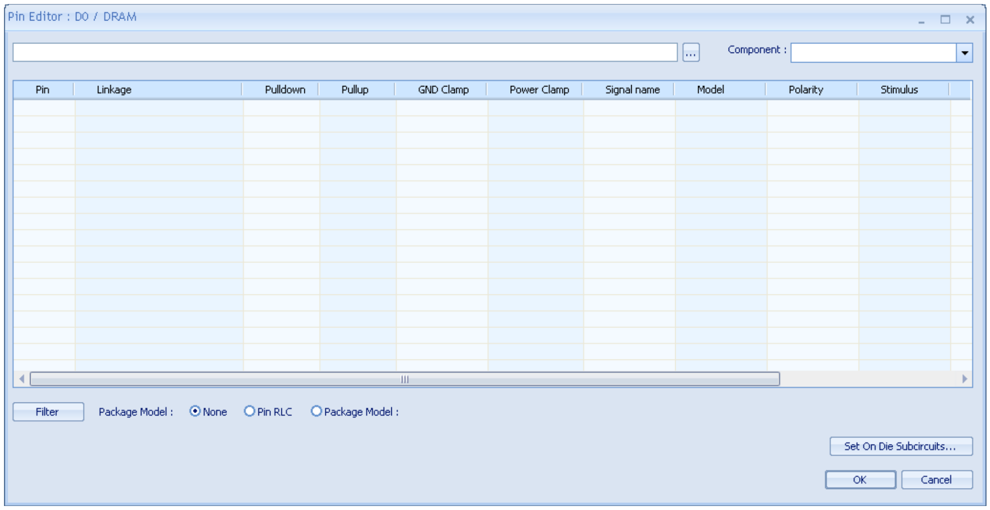


4. Select **IBIS- I/O Buffer Information Specification** from the **Model File Type** drop-down menu.

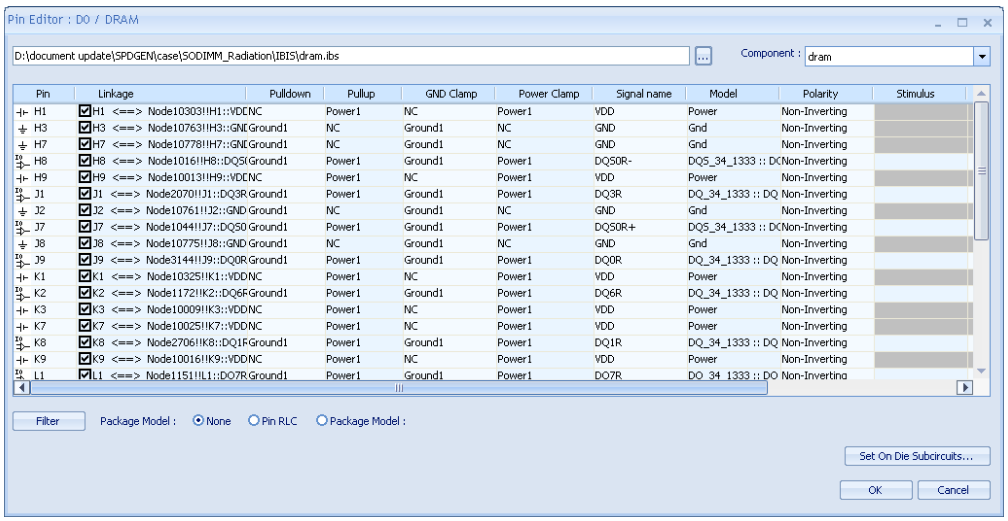


5. Click the **Edit IBIS** button.

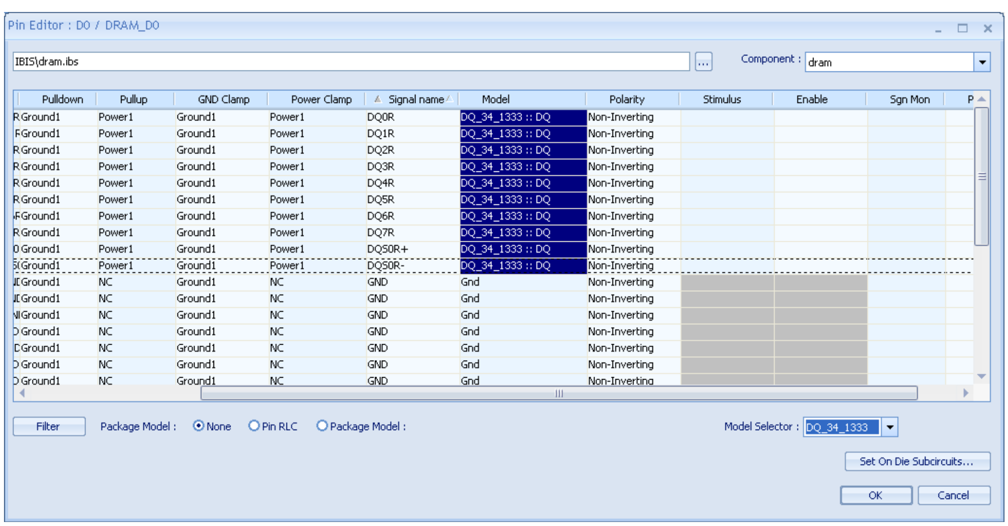
   The **Pin Editor: D0 / DRAM** window opens.



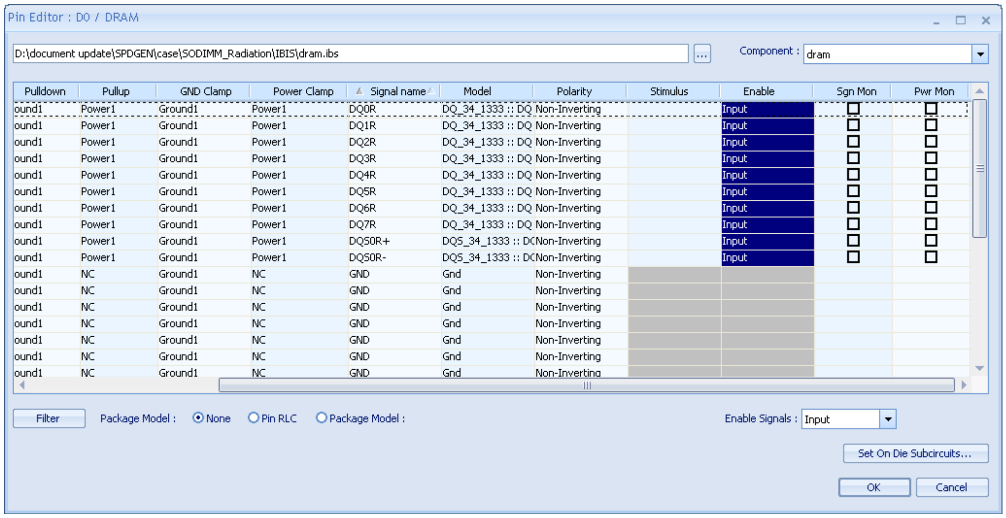
6. Click the browse button and select the file dram.ibs for **D0**.



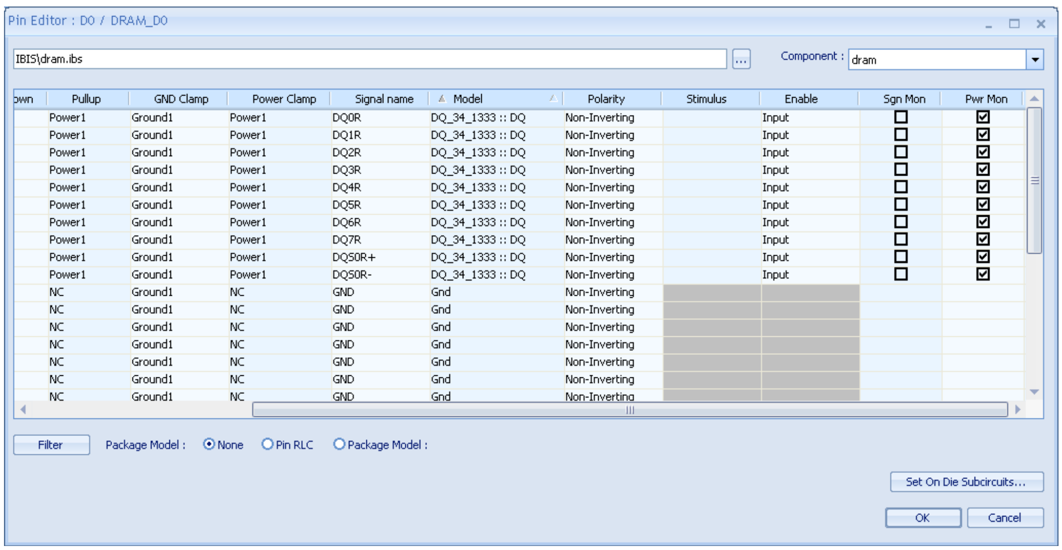
7. Select signals **DQ0R~DQ7R**, **DQS0R+**, and **DQS0R-**.
8. Select **DQ_34_1333::DQ** in the column of **Model**.



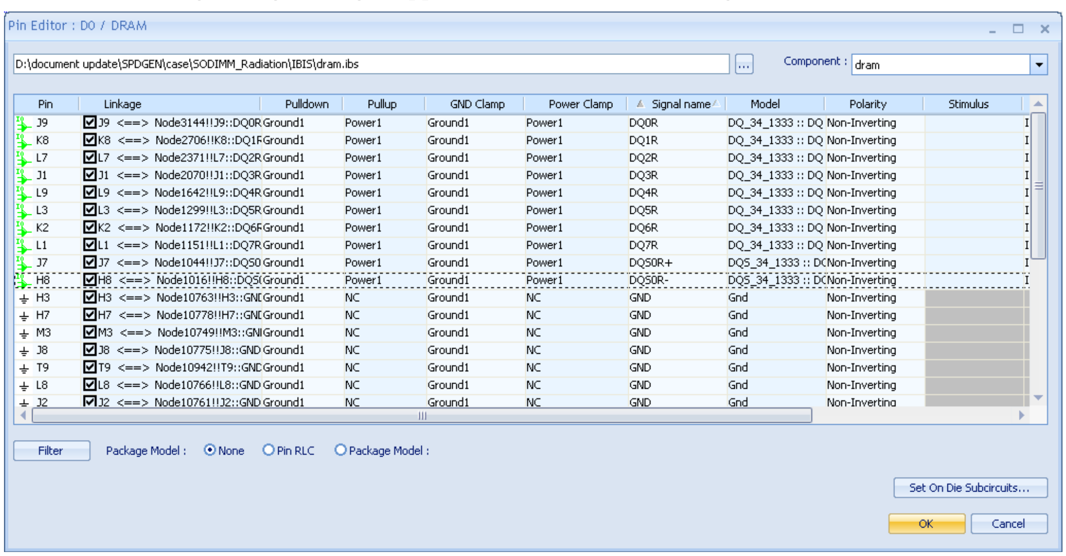
9. Select **Input** in the column of **Enable**.



10. Check **Pwr Mon**.



After all the settings, the green signs appear ahead of the selected signals.

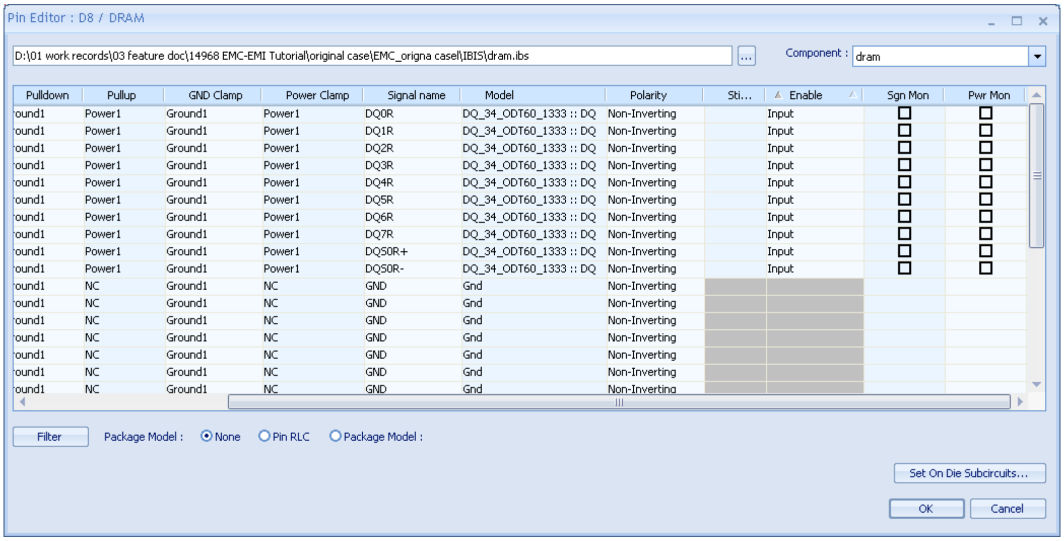


11. Click **OK**.

12. Repeat Step **2** to 11 to define the file dram.ibs for **D8** and set parameters as the following:

Model: DQ_34_ODT60_1333

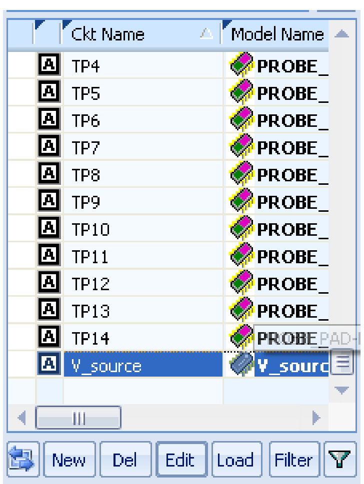
Enable: Input



3.3.3 Setting Up VRM

1. Choose Setup > Circuit/Linkage Manager….

2. Select **V _source.**



3. Click the **Edit** button.

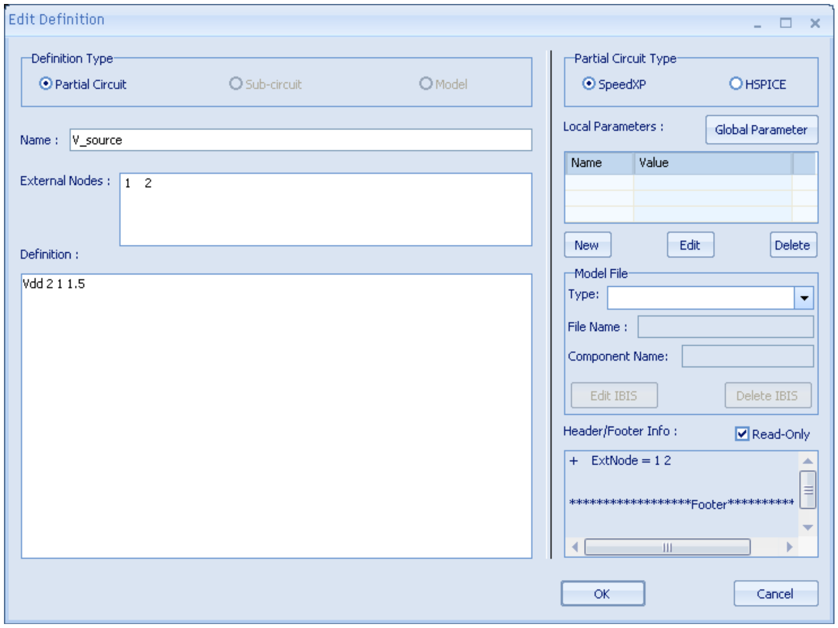
The **Edit Definition** window opens.

4. Input the definition as below.

Name: V_source

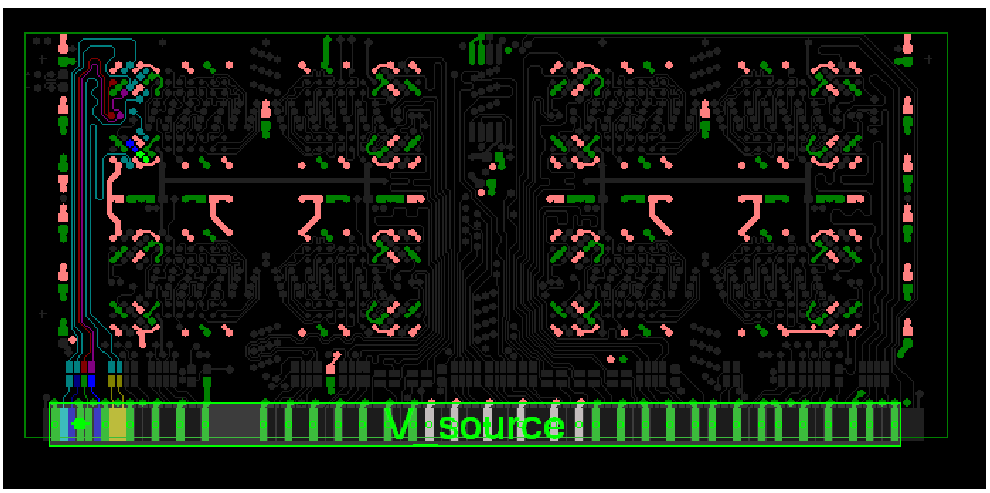
External Nodes: 1 2

Definition: Vdd 2 1 1.5



5. Click **OK**.

   V_Source is added as shown below.

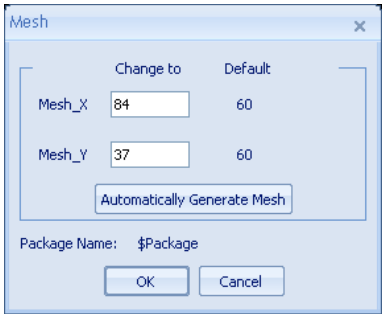


3.4 Generating Mesh

This section describes how to set the mesh for the FDTD plane solver process. The mesh can be auto-generated following the instructions below.

1. Click **Generate Mesh** in the **Workflow** pane.

   The **Mesh** window opens.



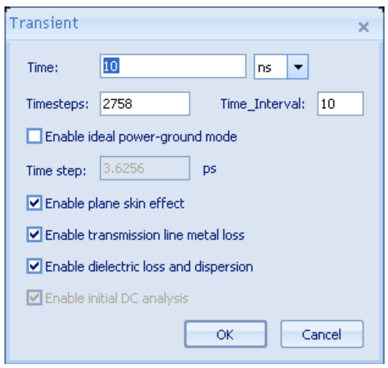
2. Click the **Automatically Generate Mesh** button.
3. Click **OK**.

3.5 Assigning Simulation Time

This section describes how to set the simulation time to the desired span.

1. Click **Assign Simulation Time** in the **Workflow** pane.

   The **Transient** window opens.



2. Input simulation time: **10ns**.
3. Check **Enable plane skin effect**, **Enable transimission line metal loss**, and **Enable dielectric loss and dispersion**.

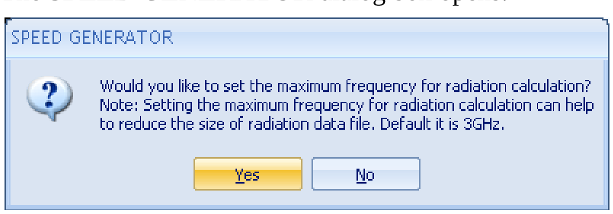
   Enable initial DC analysis is automatically checked.
4. Click **OK**.

3.6 Radiation

This section describes how to use SPEED2000 to save the data needed to do radiation post-processing. SPEED2000 allows you to do wideband radiation post-processing. However, for most practical purposes, it is only optional to generate radiation results up to a few GHz. It is necessary to set a max frequency in order to reduce the storage space needed for the post-processing.

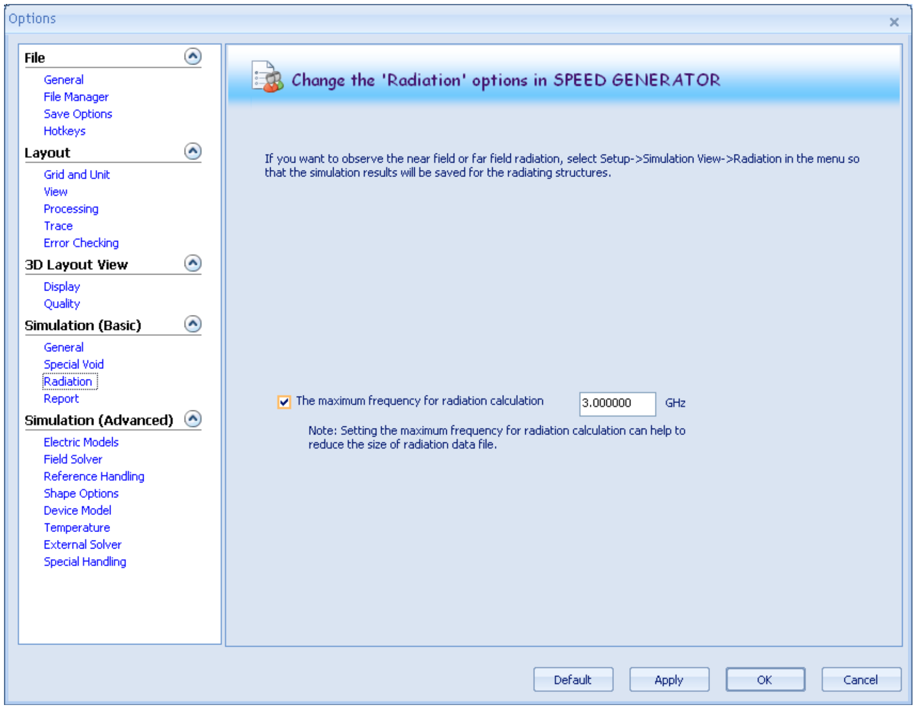
1. Click **Radiation** in the **Workflow** pane.

   The **SPEED GENERATOR** dialog box opens.

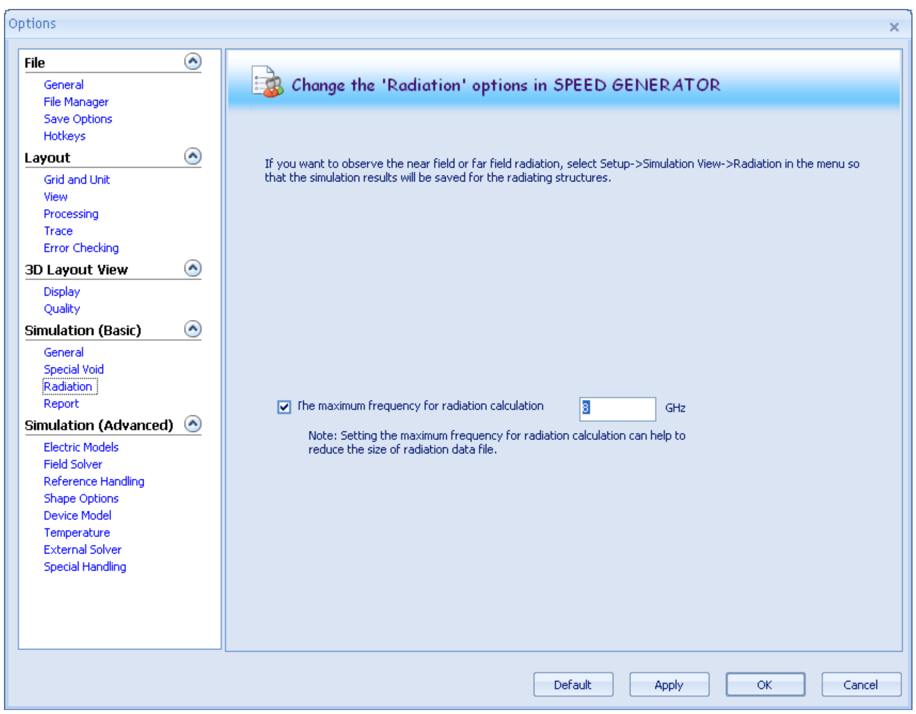


2. Click **Yes**.

The **Options** > **Simulation (Basic)** > **Radiation** window opens.



3. Change the maximum frequency for radiation calculation from **3**GHz to **8**GHz.



4. Click **OK**.

3.7 Checking Monitor Component Voltage

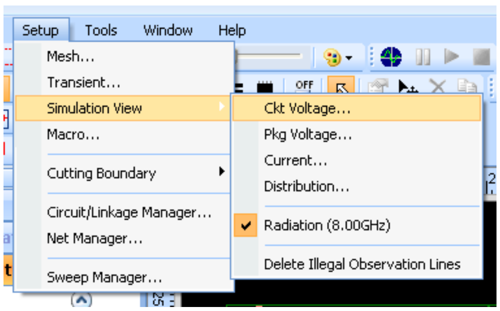
In the monitor component voltage, you can set monitor points to control the voltages that you want to measure.

You can only add probes on components that are enabled. So if you want to probe at unplaced device pins, you need to add the device model or place a large resistance across the probe points. The probe points can be used to check the conducted emissions.

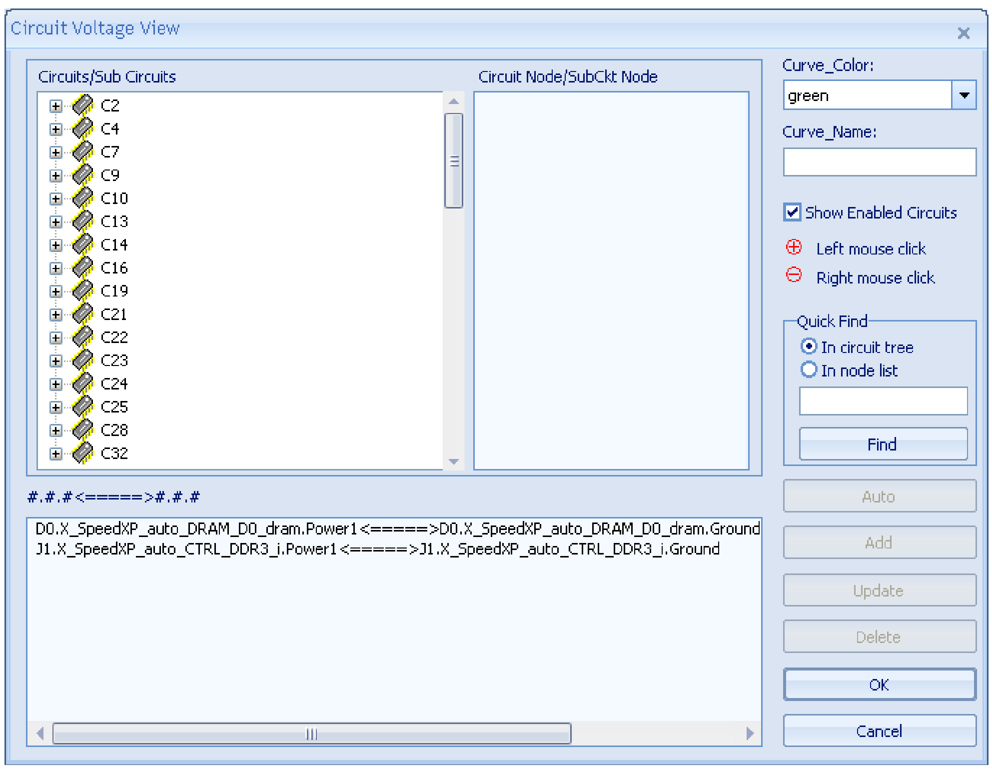
1. Click **Monitor Component Voltage** in the **Workflow** pane,

   or

   Choose Setup > Simulation View > Ckt Voltage….



   The **Circuit Voltage View** window opens.

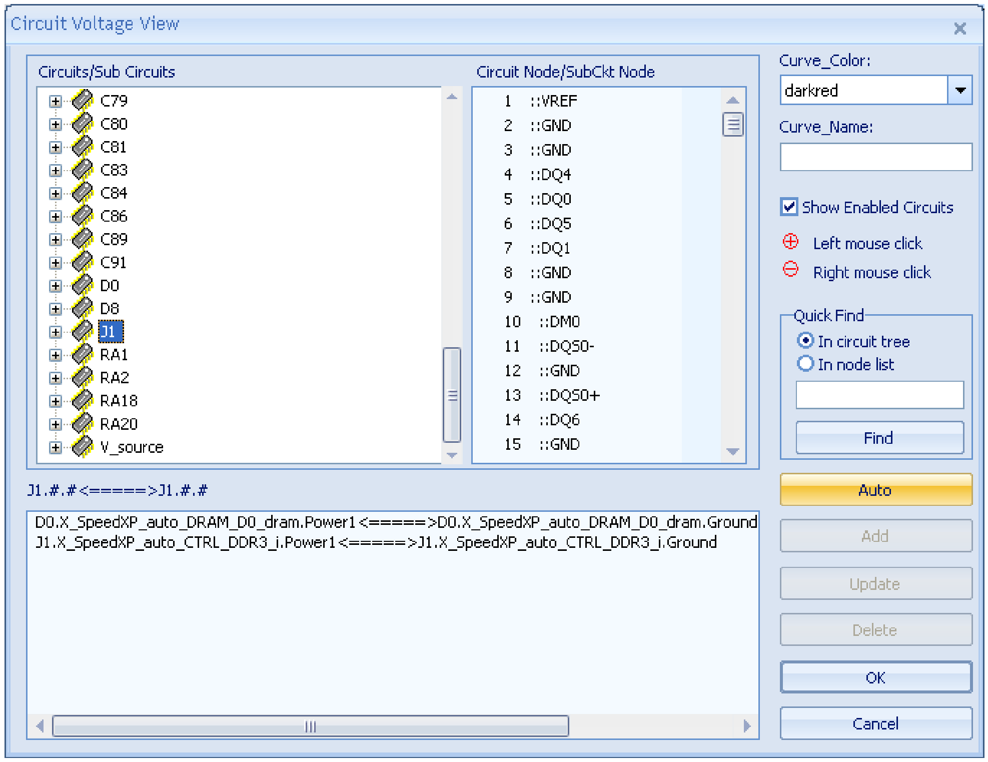


NOTE!	**J1** and **D0 Pwr Mon** are checked, so the two items appear in **Circuit Voltage View**.

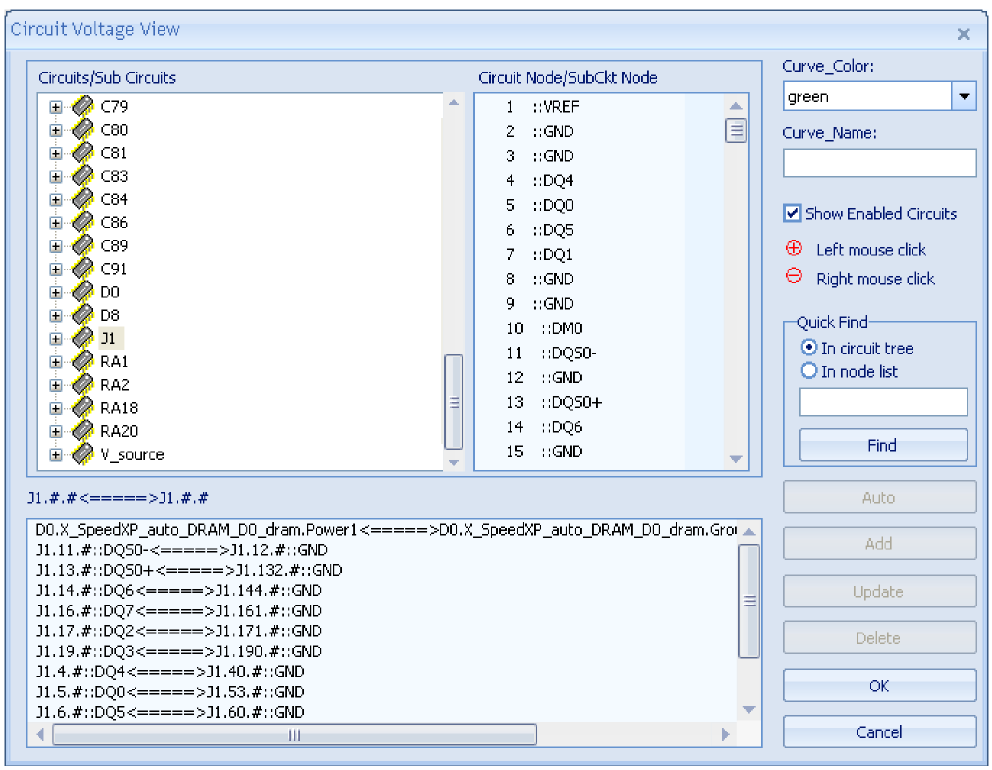
2. Add simulation view for **J1**.

   a. Click **J1** in the **Circuits/Sub Circuits** column.

b. Click the **Auto** button.



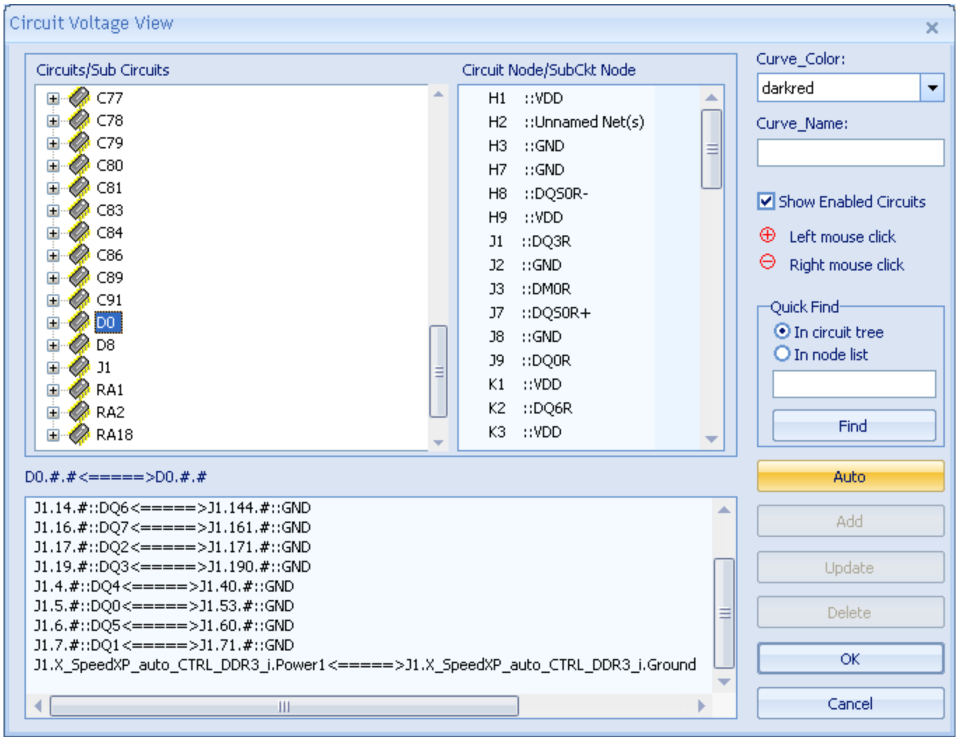
The **J1** circuit node is added automatically as below.



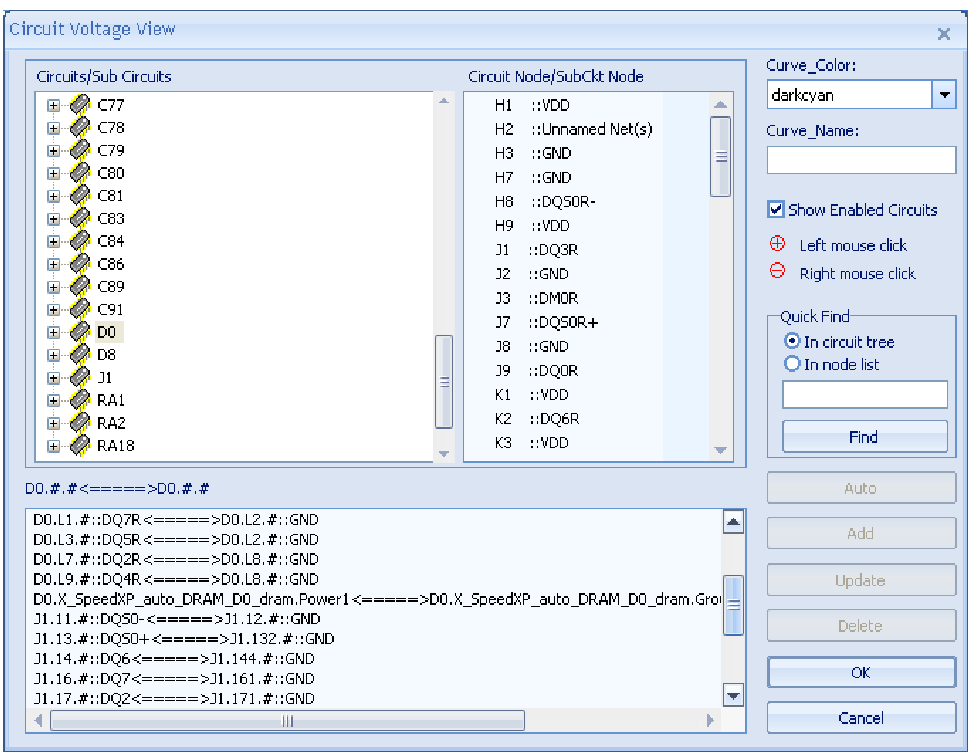
3. Add simulation view for **D0**.

a. Click **D0** in the **Circuits/Sub Circuits** column.

b. Click the **Auto** button.



The **D0** circuit node is added automatically as below.



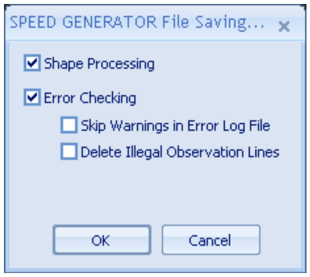
4. (Optional) Follow Step **2** and **3** to add other observations you need.
5. Check all the observed nodes and click **OK**.

4 Saving Files and Running Simulation

4.1 Saving File with Error Check

1. Click **Save File with Error Check** in the **Workflow** pane.

   The **SPEED GENERATOR File Saving Options** window opens.

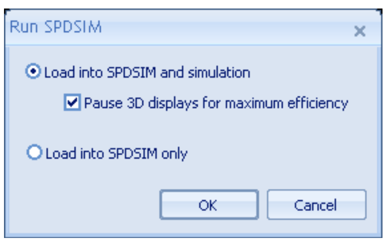


2. Click **OK**.

4.2 Running Simulation

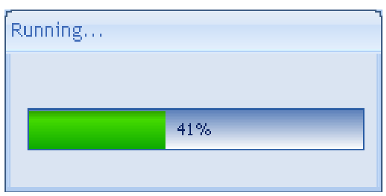
1. Click **Start simulation** in the **Workflow** pane.

   The **Run SPDSIM** window opens.



2. Click **OK**.

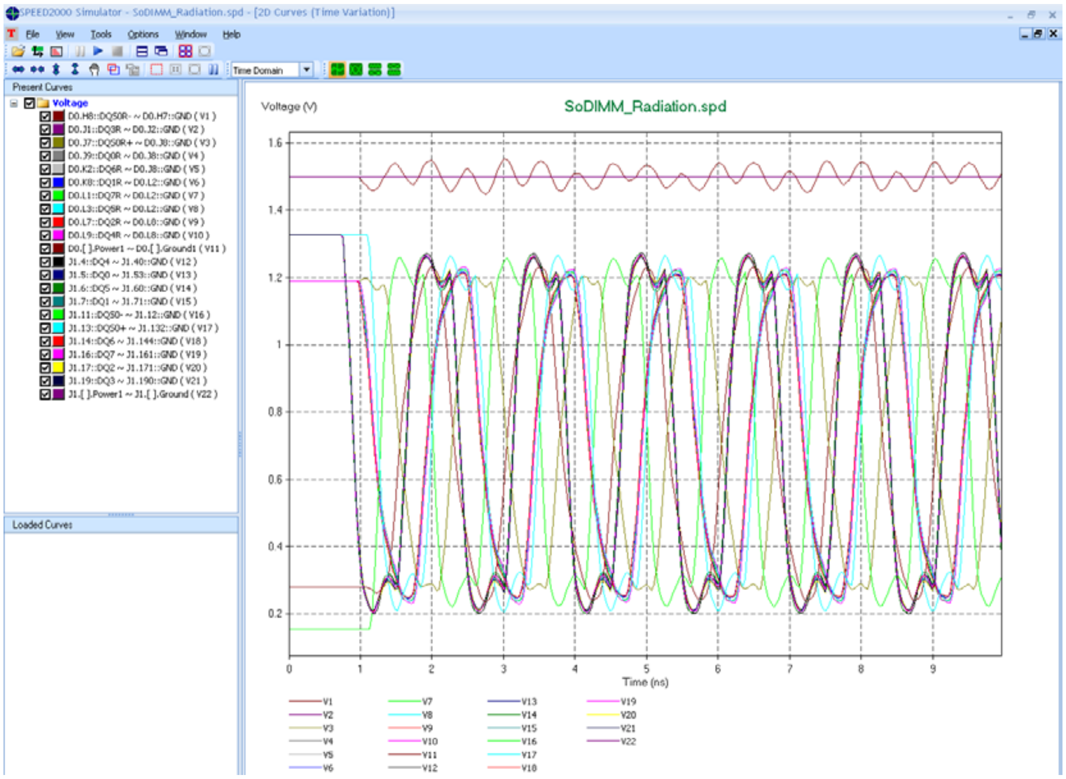
   SPEED Simulator starts to simulate. A green bar appears to show the progress of simulation.



5 Viewing Results

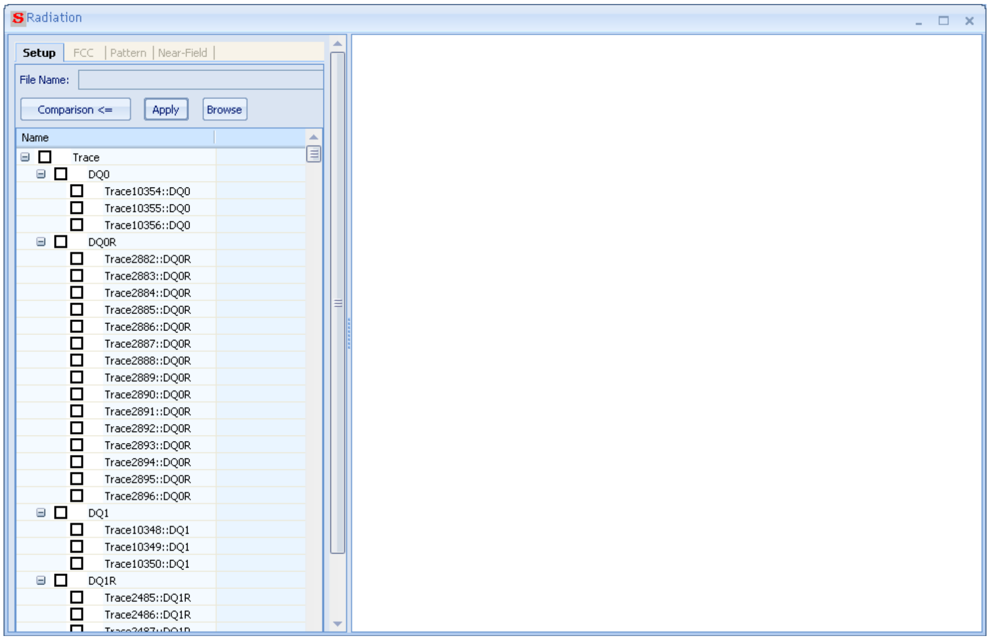
5.1 Viewing Time Domain Result

After the simulation, the 2D curves show the time domain result.

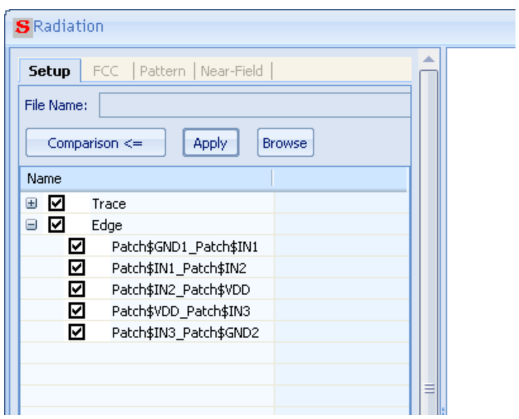


5.2 Viewing Radiation Result

After the simulation, the **Radiation** window appears to show the result.



1. Click the **Setup** tab.
2. Select all traces and edges in this tutorial to observe the radiation.

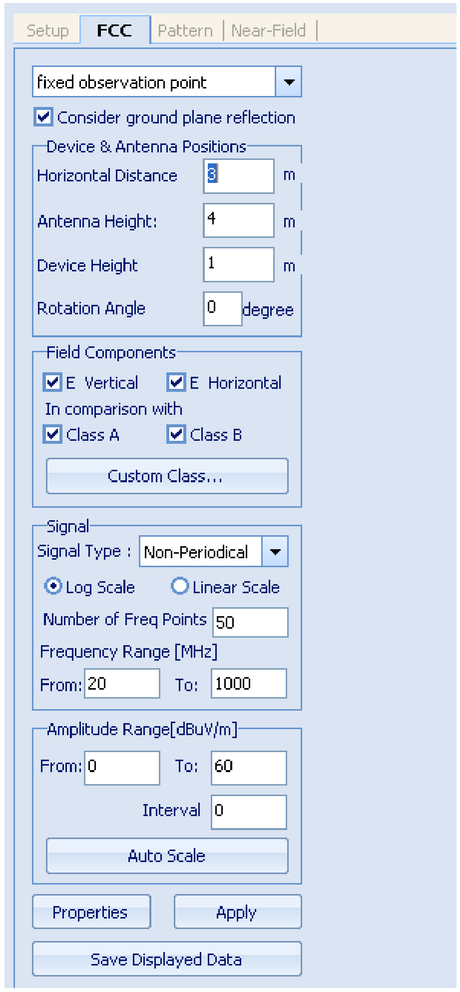


3. Click the **Apply** button to refresh the result including the **FCC**, **Pattern** and **Near-Field** results.

NOTE!	You can select any one or more traces, wirebonds, leads and edges, and click the **Apply** button to observe the radiation.

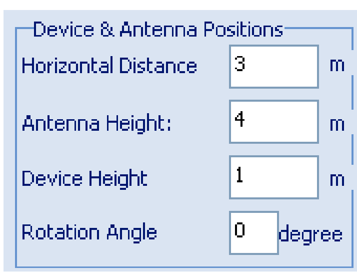
5.2.1 FCC Result

1. Click the **FCC** tab.

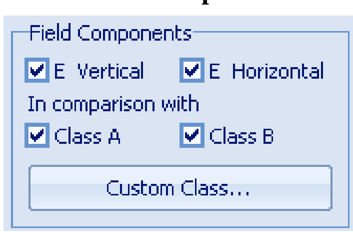


2. Define the parameters for **FCC**.

   a. Define the box size: **Antenna Height** and **Rotation Angle**.



   b. Define **Field Components**.



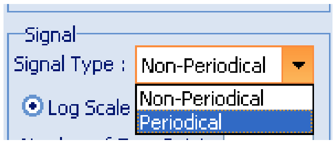
You can

- Select **E Vertical** or **E Horizontal** to view the result

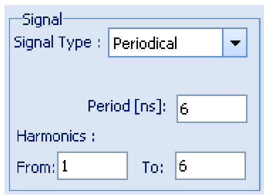
- Define class by clicking the **Custom Class...** button

c. Define **Signal Type**.

- Signal Type: **Periodical**

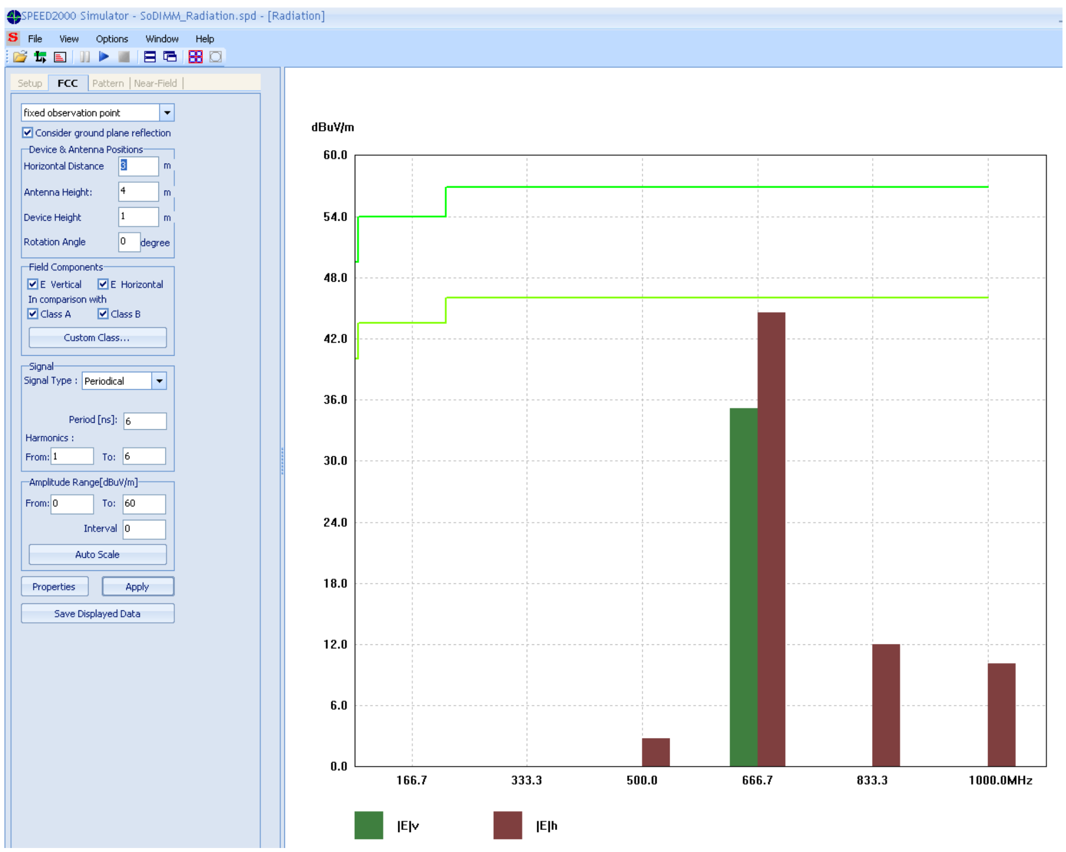


- Period[ns]**: 6**

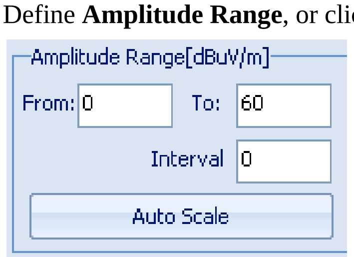


3. Click the **Apply** button.

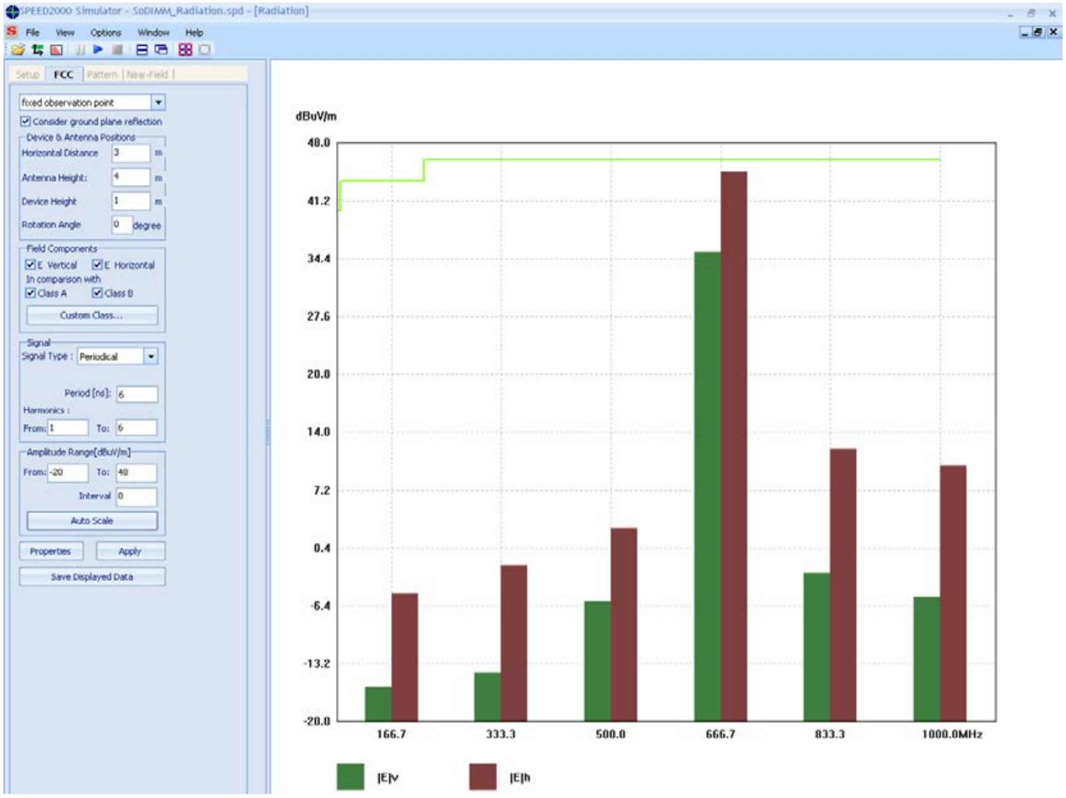
   The FCC result shows.



4. Define **Amplitude Range**, or click the **Auto Scale** button to fit the result.



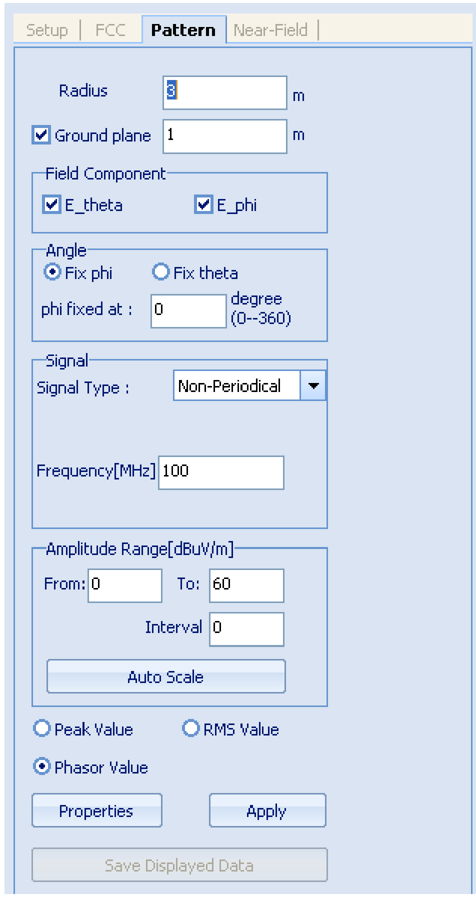
The result becomes fit as below.



5. (Optional) Move the pointer to the FCC to view the data.
6. (Optional) Click **Save Displayed Data** to save the displayed data.

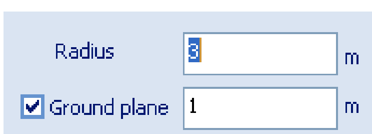
5.2.2 Pattern Result

1. Click the **Pattern** tab.



2. Define the parameters for **Pattern**.

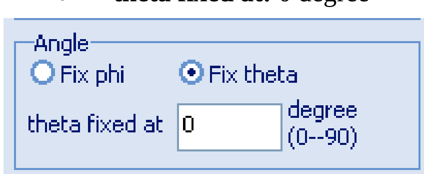
   a. Define the **Radius** value and the distance of **Ground plane**.



NOTE!	If you enable the ground plane, there will be a ground plane in the Z-axis.

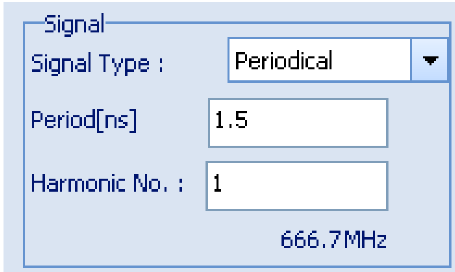
   b. Define the pattern form.

   - **Angle:** Fix theta
   - **theta fixed at:** 0 degree



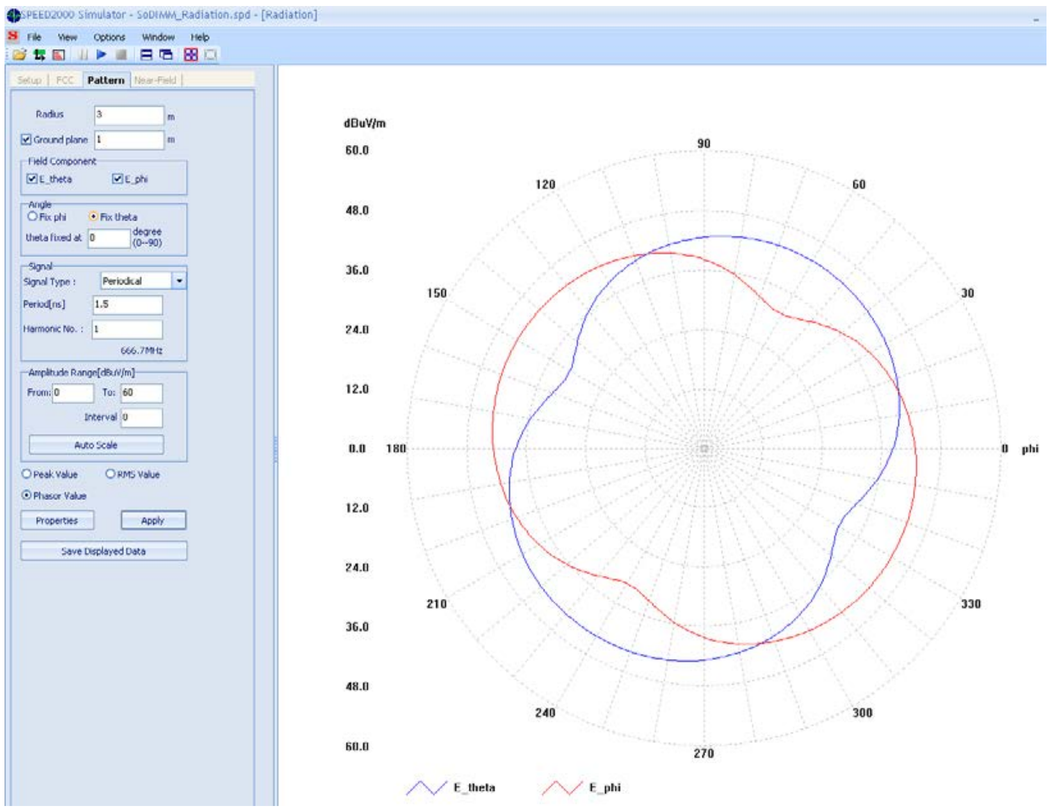
   c. Define **Signal Type**.

   - **Signal Type:** Periodical
   - **Period[ns]:** 1.5

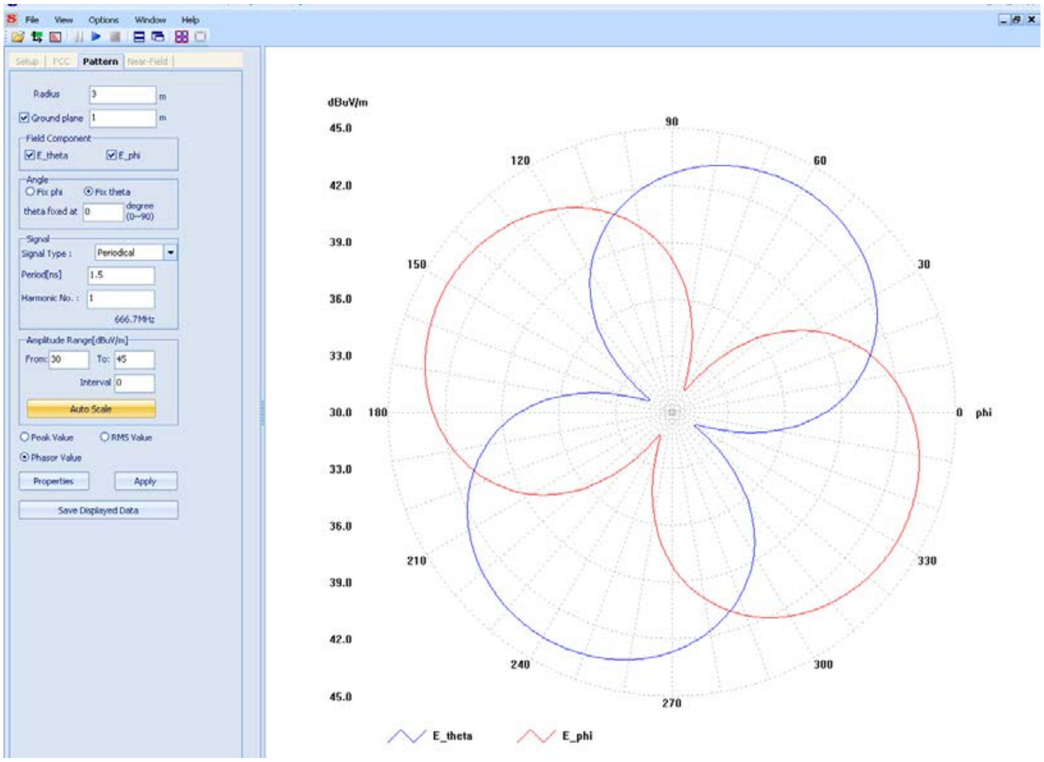


3. Click the **Apply** button.

   The pattern result shows.



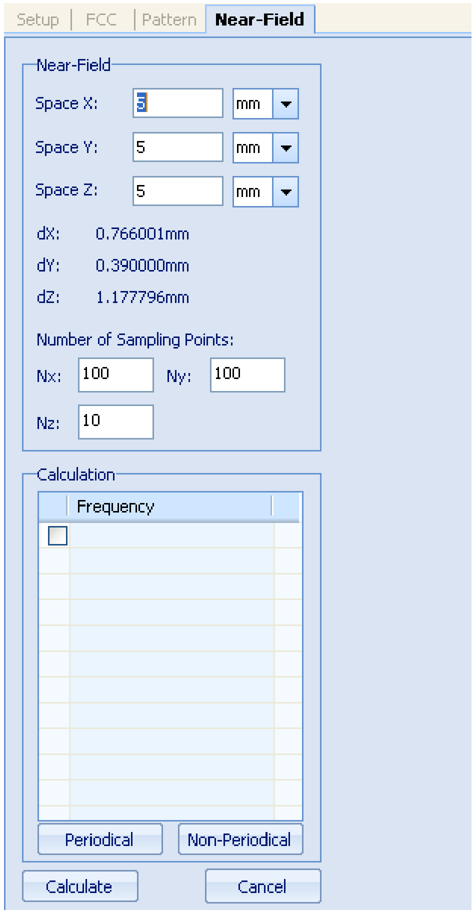
4. Define the **Amplitude Range**, or click the **Auto Scale** button to fit the result.



5. (Optional) Move the pointer to the pattern to view the data.
6. (Optional) Click the **Save Displayed Data** button to save the display data.

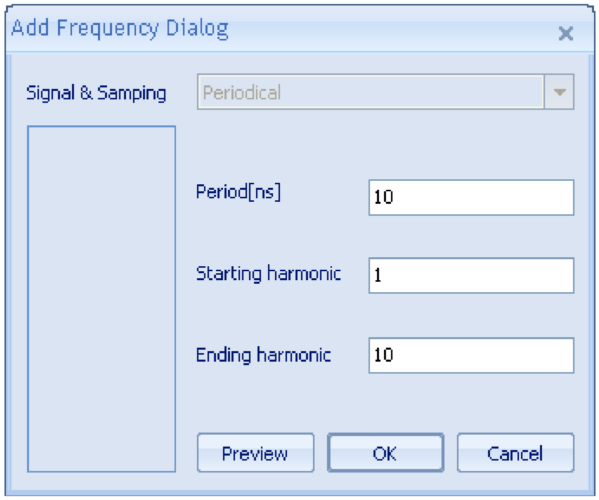
5.2.3 Near-Field Result

1. Click the **Near-Field** tab.



2. Define the parameters for **Near-Field**.
   a. Set the Near-Field space.
      - Space X: 5mm
      - Space Y: 5mm
      - Space Z: 5mm
   b. Define the number of sampling points.
      - Nx: 100
      - Ny: 100
      - Nz : 10
3. Define calculation.
   a. Click the **Periodical** button.

      The **Add Frequency Dialog** window opens.

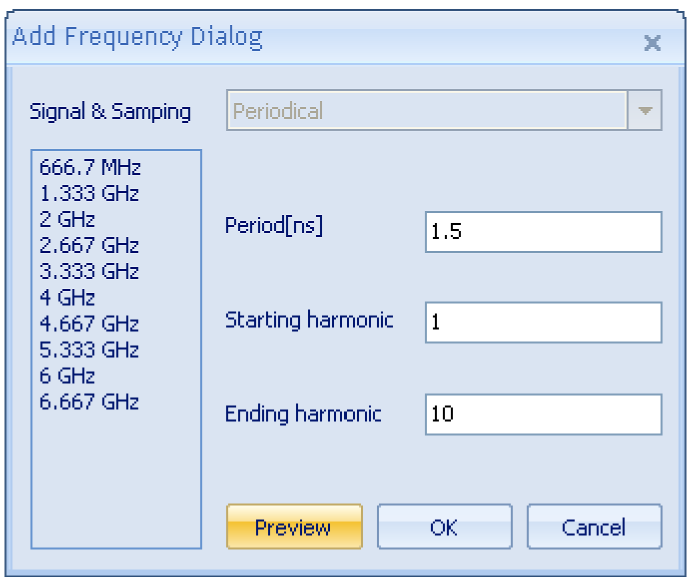


b. Set the following parameters:

- **Period[ns]:** 1.5
- **Starting harmonic:** 1
- **Ending harmonic: 10**

c. Click the **Preview** button.

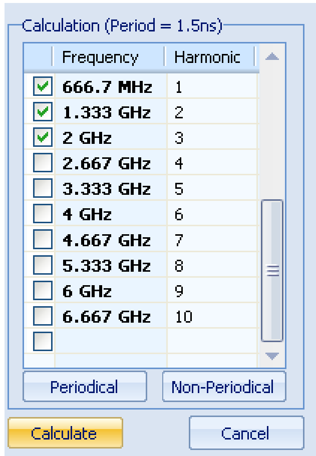
The frequencies show in the left field.



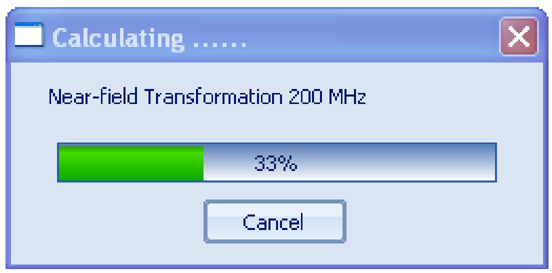
d. Click **OK**.

The frequency shows in the **Calculation** column.

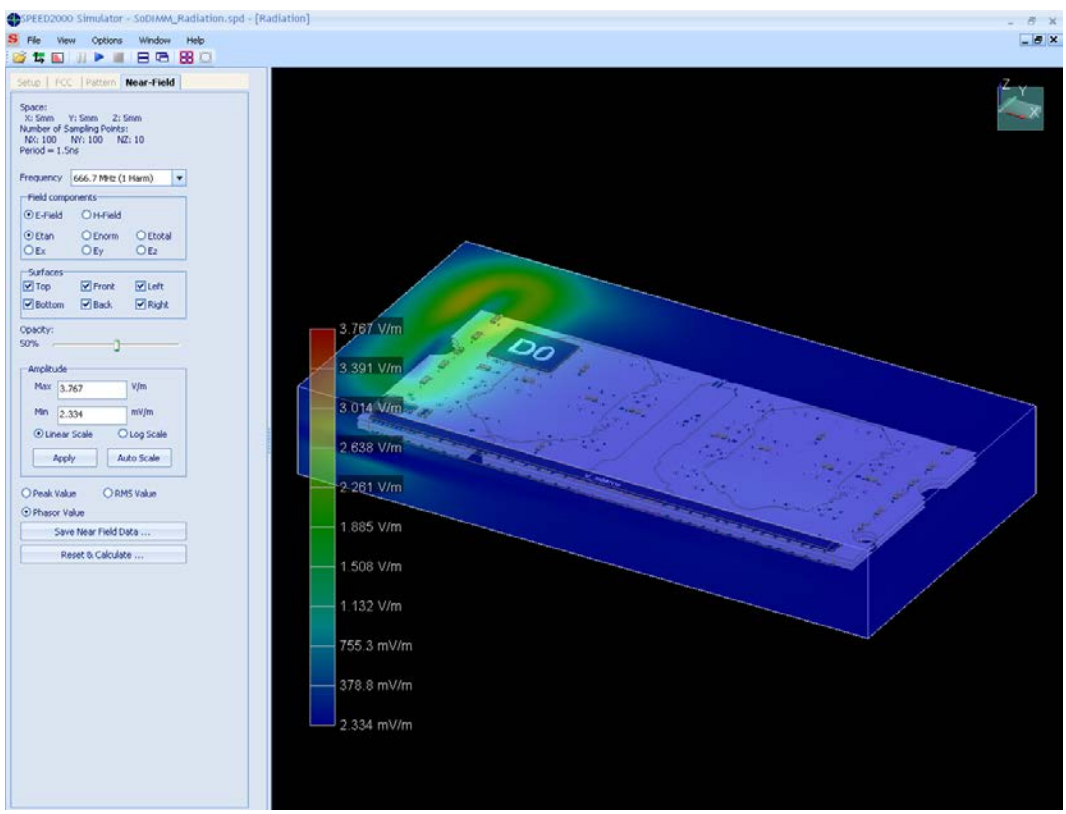
4. Select three frequency points, and click the **Calculate** button.



A green bar appears showing the progress of calculation.



After the calculation, the near-field result shows.



5. (Optional) You can change the **Frequency**, **Field components**, **Surfaces** and **Amplitude** to observe the related results.

6. (Optional) Move the pointer to the field to view the data.
7. (Optional) Click the **Save Near Field Data ...** button to save the data.