cadence®

General_SI_Simulation_Tutorial_2

Product Version 16.6
July 2014

Legal

Cadence Design Systems, Inc. (Cadence), 2655 Seely Ave., San Jose, CA 95134, USA.

Contents

1 4
Introduction 4
Overview 4
Tutorial Summary 5
2 9
Layout Setup 9
3 11
Simulation Setup 11
Enabling Simulation Nets 11
Showing Aggressor Nets 12
Assigning DC Voltage 14
Assigning Component Models 15
Assigning IBIS Model to IC Component U0 16
Assigning IBIS Model to IC Component U1 20
Copying IBIS Model to U2, U3 and U4 from U1 22
SI Simulation Options Setup 23
General SI Options 24
U0 Parameters Setup 25
U1 Parameters Setup 26
Probes Setup 27
Saving File 27
4 29
Simulation and Results 29

1

Introduction

This tutorial introduces how to perform simulation with SPEED2000 **General SI Simulation (Ideal P/G)** workflow.

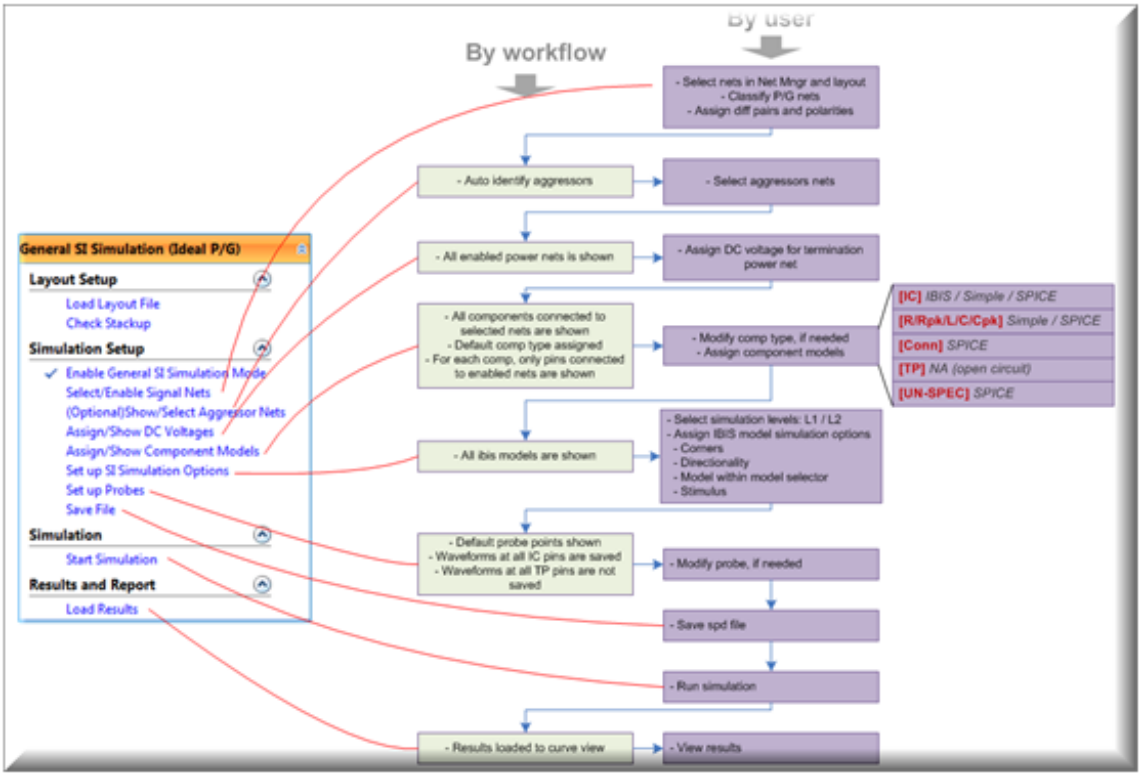
Overview

The SPEED2000 **General SI Simulation (Ideal P/G)** workflow is for general purpose signal integrity simulations without considering the non-ideal power/ground effects. Specifically,

- Level 1 – Trace and via couplings not considered, ideal P/G
- Level 2 – Trace and via couplings considered, ideal P/G

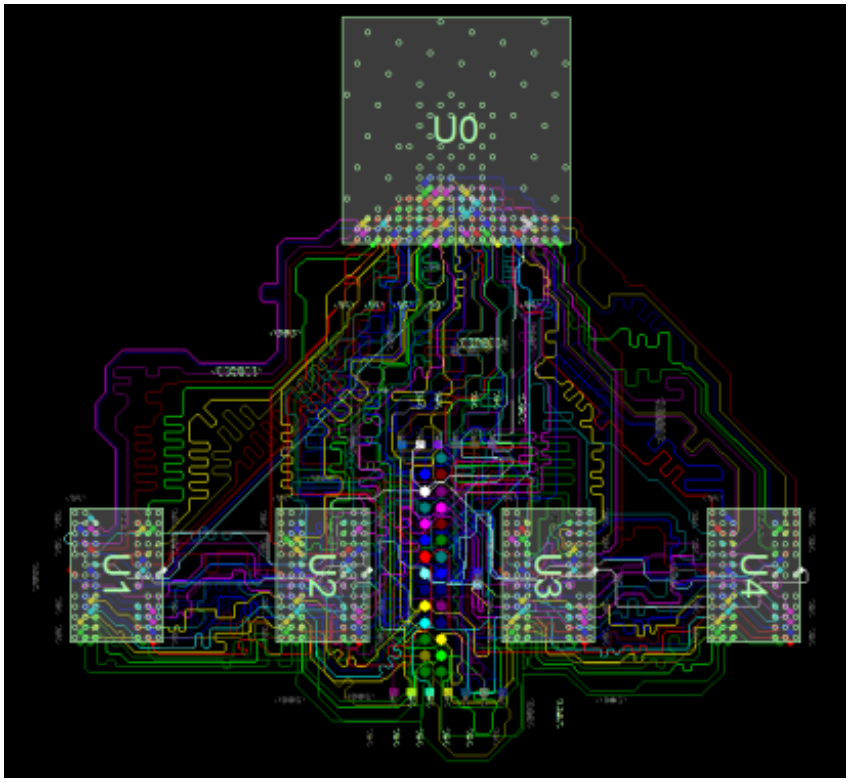
In General SI Simulation (Ideal/P/G) workflow, you can firstly enable signal nets to be simulated, which are referred as primary nets. In order to better simulate xtalk effects, a new feature - to select which aggressor nets to be included in simulation (optional) - is added to find all the aggressor nets through trace coupling and/or via coupling to primary nets. The primary nets and selected aggressor nets are all included in the subsequent model setup, simulation option setup and simulations. By default, only waveforms at IC components for primary nets are saved.

The main steps for simulation setup are shown in the following block diagram.

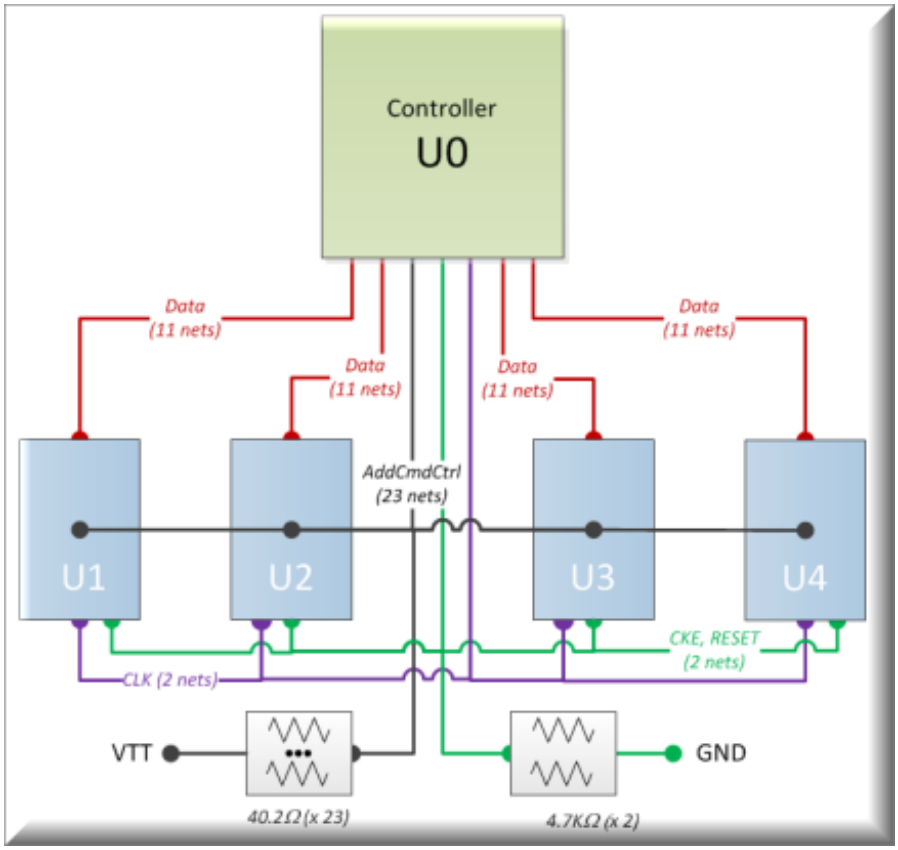


Tutorial Summary

The sample case used in this tutorial contains one controller and four DRAMs as shown below.



The topology is as the following figure shows.



In this tutorial, only nets **BA0** and **DQ1** are included in analysis as primary nets. Aggressors to these primary nets are automatically shown with user-defined coupling criteria.

The following three original files are used in this tutorial:

- Tutorial_GSI.spd – layout file
- dram_IO.ibs – IBIS file containing DRAM buffer models
- ctrl_IO.ibs – IBIS file containing Ctrl buffer models
- They are all located in: `<INSTALL_DIR>\SpeedXP\Samples\SPEED2000\General SI Simulation\Examples_PreSetup\`

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\General SI Simulation \Examples_PostSetup\Tutorial-2\`

General SI Simulation (Ideal P/G) workflow will lead user to:

- Setup simulation parameters

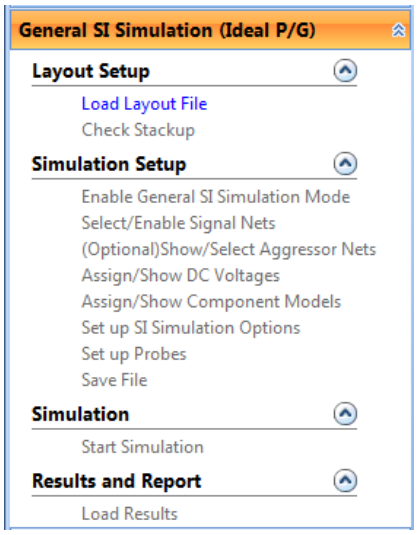
- Run simulation
- View simulation results

2

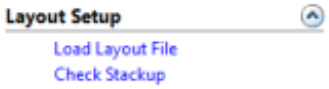
Layout Setup

This chapter describes how to load layout file and check stackup.

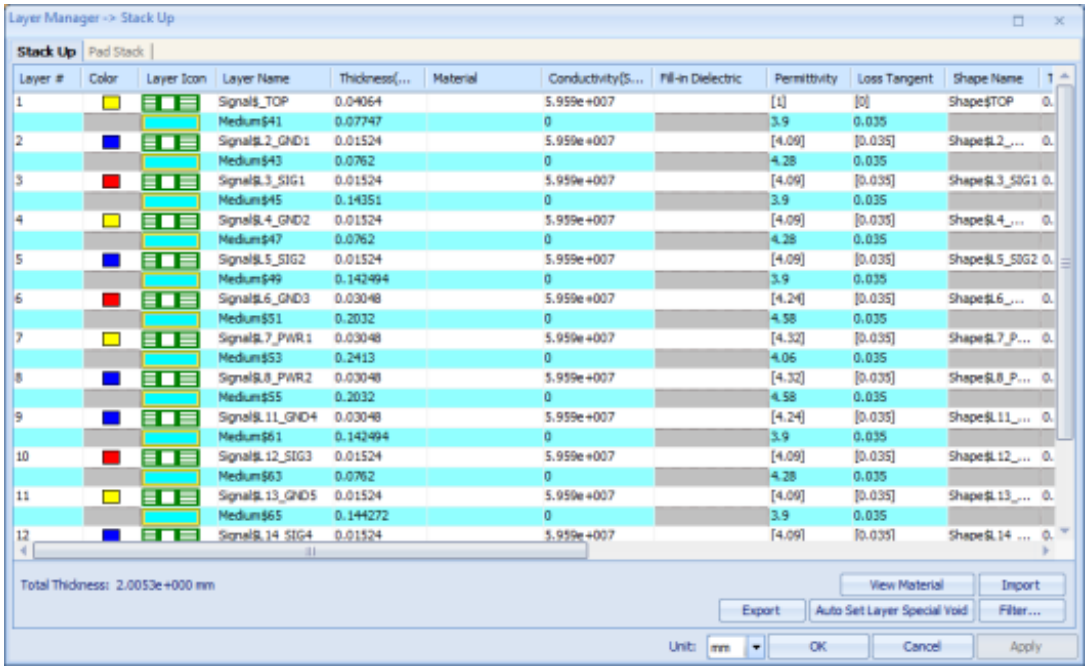
1. Launch **SPEED2000 Generator** , and choose the **General SI Simulation (Ideal P/G)** workflow.



2. Click **Load Layout File** to load tutorial_GSI.spd.
   The workflow step **Check Stackup** is enabled.



3. Click **Check Stackup** to open the **Layer manager -> Stack Up** window.



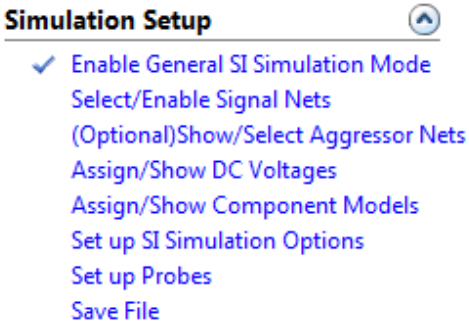
4. Check the stackup and edit as desired (no changes in this example).
5. Click **OK** to quit the window.

3

Simulation Setup

This chapter introduces how to set up simulation parameters for general SI simulation.

Click **Enable General SI Simulation Mode** to enable General SI simulation mode.

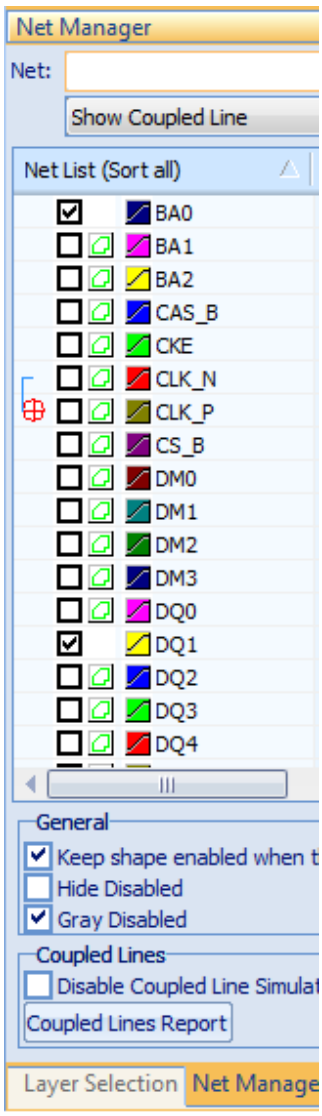


When enabled, a check mark ✓ appears ahead the workflow step. And all other related steps are enabled.

Enabling Simulation Nets

To perform general SI simulation with SPEED2000, you'll need to select simulation nets first and define the differential pair property.

1. Click **Select/Enable Signal Nets** in the **Workflow** pane.
   The **Net Manager** appears in the right side of the window.
2. Click to enable power net **VTT** and signal nets **BA0** and **DQ1** .



Icon

The diff-pair and polarity are guessed out automatically.

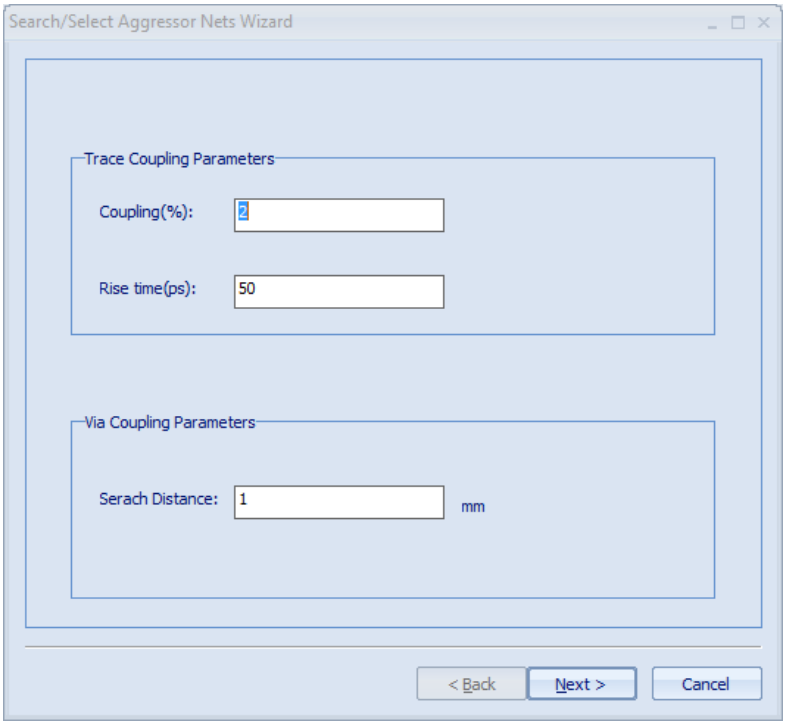
- If the diff-pair is not guessed out, choose both positive and negative nets, right-click and choose **Classify as diff pair** to define it manually.
- If the polarity is not guessed out, right-click the positive net and choose **Switch Polarity** to define it manually.

Showing Aggressor Nets

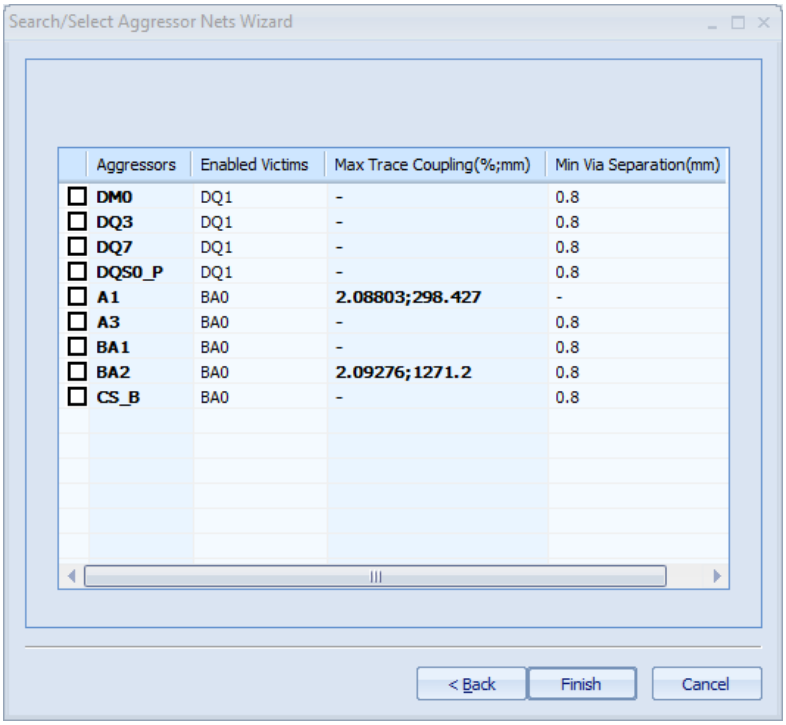
This step is to identify potential aggressors for BA0 and DQ1 based on trace coupling and via

coupling. You can manually choose the desired aggressor signal nets to be included in simulation.

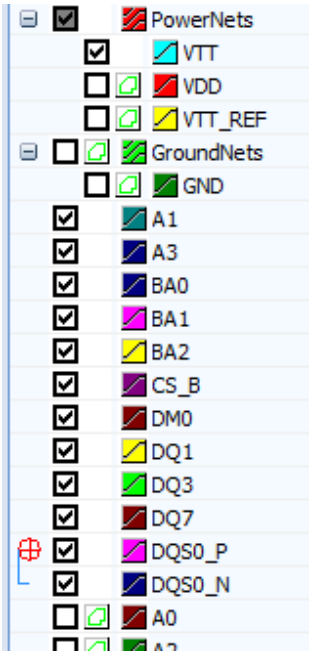
1. Click **Show/Select Aggressor Nets** in the **Workflow** pane.
   The **Search/Select Aggressor Nets Wizard** opens.
2. Input the following values to each field:
   - **Coupling(%):** 2
   - **Rise time(ps):** 50
   - **Search Distance:** 1 (mm)



3. Click **Next** .
   SPDGEN starts to enable nets and analyze aggressor nets. The time it takes depends on the size and settings of the specific case. When the process is completed, the **Search/Select Aggresssor Nets Wizard** displays all aggressors nets.



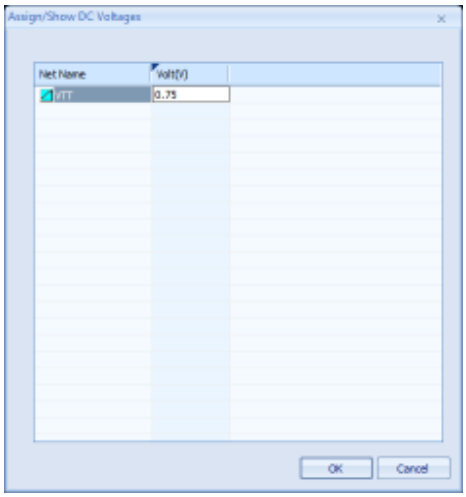
4. Click to enable all nets in this sample and click **Finish** .
   The **Search/Select Aggressor Nets Wizard** exists and the corresponding nets are enabled in **Net Manager** .



Assigning DC Voltage

If power or ground nets are enabled as simulation nets, the DC voltage should be assigned before simulation.

1. Click **Assign/Show DC Voltages** in the **Workflow** pane.
   The **Assign/Show DC Voltages** window opens.
2. Input **0.75V** for **VTT** net like the following figure shows.



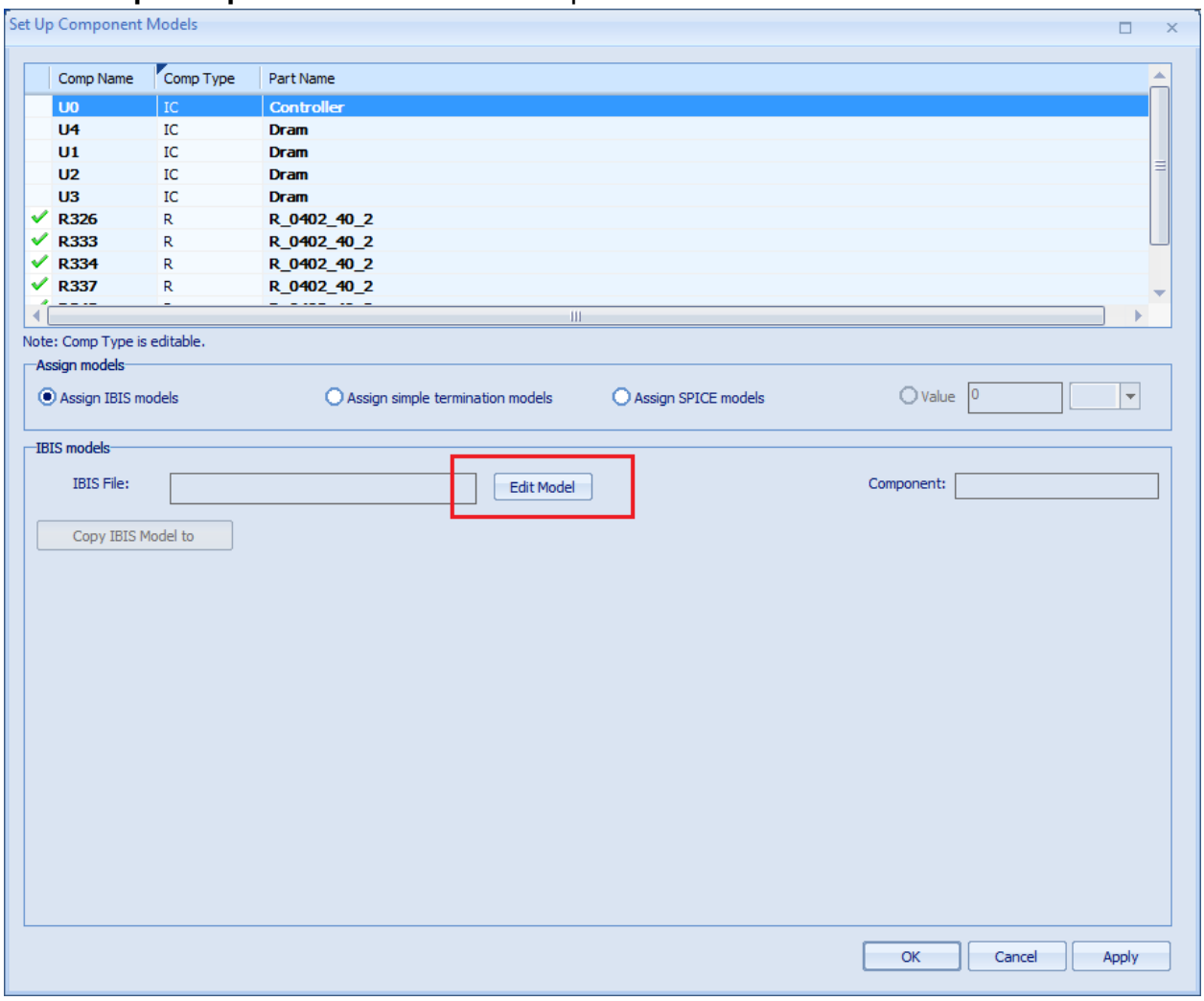
3. Click **OK** .

Assigning Component Models

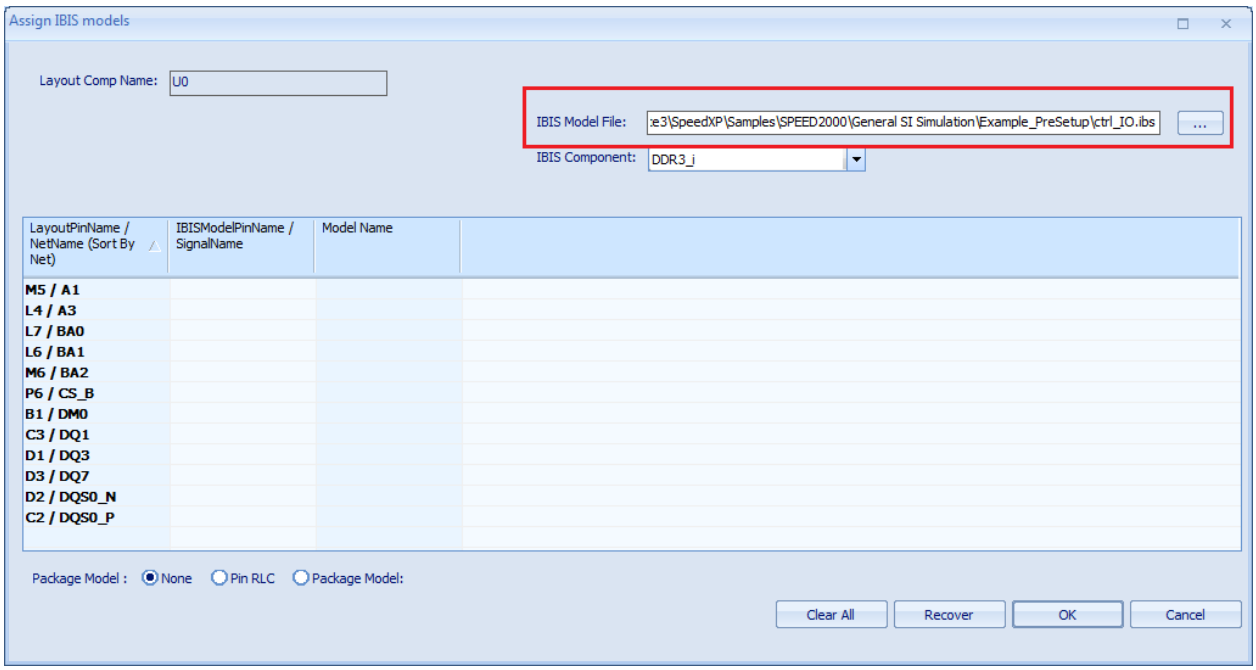
This section introduces how to assign models to all components.

Assigning IBIS Model to IC Component U0

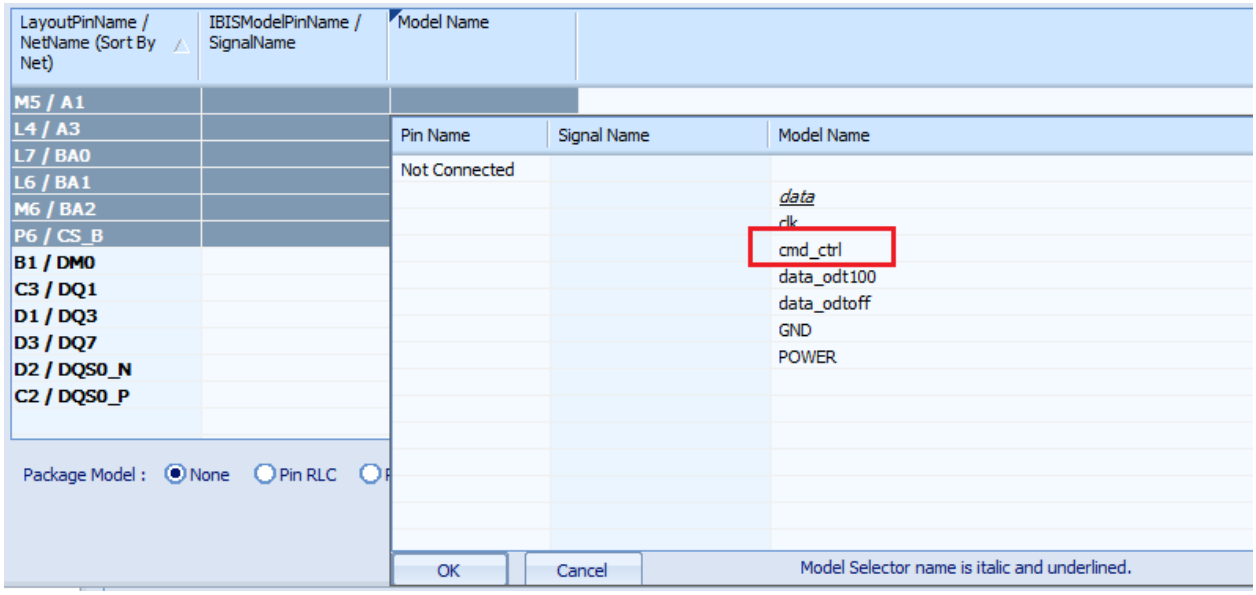
1. Click **Assign/Show Component Models** in the **Workflow** pane.
   The **Set up Component Models** window opens.



2. Select to highlight component **U0** and click the **Edit Model** button.
   The **Assign IBIS models** window opens.



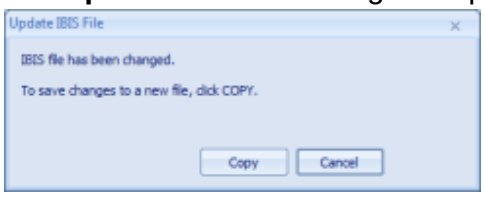
3. Browse to load the IBIS file ctrl_IO.ibs.
   The IBIS file is loaded.
4. Use **Shift** and **Ctrl** keys to select multiple nets **A1** – **CS_B** .
5. Click the icon in the field of **Model Name** column and select **cmd_ctrl** from the pop-up window.



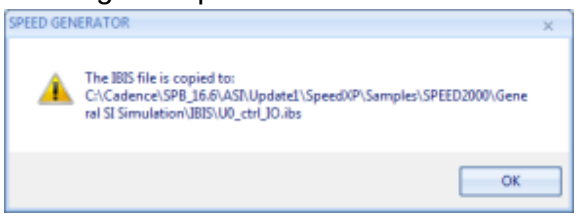
6. Click **OK** .
   The selected nets are assigned with the model **cmd_ctrl** .
   Repeat the above steps to assign nets **DM0** – **DQS0_P** with the model **data** .
   All nets are assigned with IBIS models.

LayoutPinName / NetName (Sort By Net)	IBISModelPinName / SignalName	Model Name
M5 / A1	M5 / A1	**cmd_ctrl**
L4 / A3	L4 / A3	**cmd_ctrl**
L7 / BA0	L7 / BA0	**cmd_ctrl**
L6 / BA1	L6 / BA1	**cmd_ctrl**
M6 / BA2	M6 / BA2	**cmd_ctrl**
P6 / CS_B	P6 / CS_B	**cmd_ctrl**
B1 / DM0	B1 / DM0	***data***
C3 / DQ1	C3 / DQ1	***data***
D1 / DQ3	D1 / DQ3	***data***
D3 / DQ7	D3 / DQ7	***data***
D2 / DQS0_N	D2 / DQS0_N	***data***
C2 / DQS0_P	C2 / DQS0_P	***data***

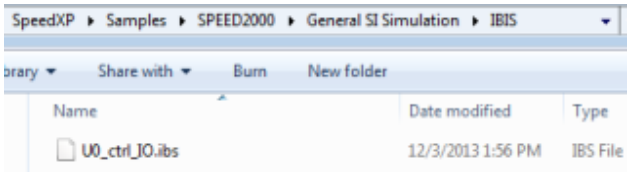
7. Check the newly generated models and click **OK** .
   The **Update IBIS File** dialog box opens to confirm the changes.



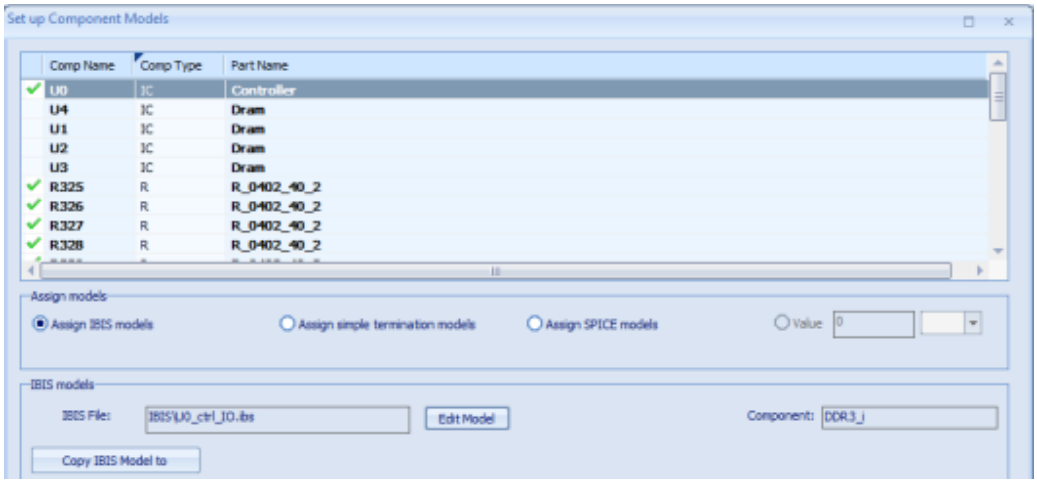
8. Click **Copy** .
   A dialog box opens to confirm the location where the IBIS file is copied to.



9. Click **OK** to create a new IBIS model.
   The **Assign IBIS models** window quits and a pin matched component IBIS model for controller **U0** is generated, located in the sub-folder named **IBIS** under the project folder.



   A green check mark appears ahead **U0** in the **Set up Component Models** window.

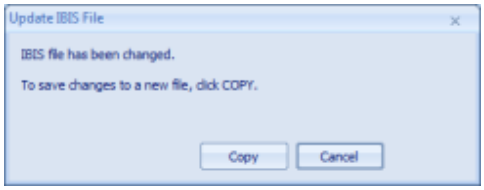

Set up Component Models
Comp Name
Comp Type
Part Name
U0
IC
Controller
U4
IC
Dram
U1
IC
Dram
U2
IC
Dram
U3
IC
Dram
R325
R
R_0402_40_2
R326
R
R_0402_40_2
R327
R
R_0402_40_2
R328
R
R_0402_40_2
Assign models
Assign IBIS models
Assign simple termination models
Assign SPICE models
Value
IBIS models
IBIS File:
IBIS\U0_ctrl_IO.ibs
Edit Model
Component:
DDR3_i
Copy IBIS Model to

Assigning IBIS Model to IC Component U1

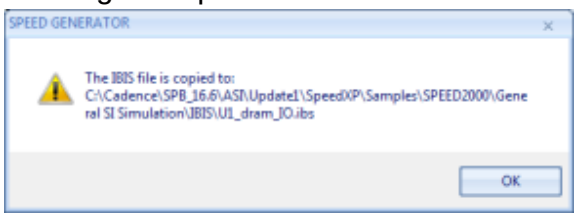
1. Select to highlight component **U1** and click the **Edit Model** button.
   The **Assign IBIS models** window opens.
2. Browse to load the IBIS file dram_IO.ibs.
   The IBIS file is loaded.
3. Repeat the steps described in the above section to assign nets **A1** – **CS_B** with the model **INPUT** .
4. Assign nets **DM0** – **DQ7** with the model **DQ** .
5. Assign net **DQS0_N** and **DQS0_P** with the model **DQS** .
   All nets are assigned with the IBIS models.

LayoutPinName / NetName (Sort By Net)	IBISModelPinName / SignalName	Model Name
L7 / A1	L7 / A1	***INPUT***
K2 / A3	K2 / A3	***INPUT***
J2 / BA0	J2 / BA0	***INPUT***
K8 / BA1	K8 / BA1	***INPUT***
J3 / BA2	J3 / BA2	***INPUT***
H2 / CS_B	H2 / CS_B	***INPUT***
B7 / DM0	B7 / DM0	***DQ***
C7 / DQ1	C7 / DQ1	***DQ***
C8 / DQ3	C8 / DQ3	***DQ***
E7 / DQ7	E7 / DQ7	***DQ***
D3 / DQS0_N	D3 / DQS0_N	***DQS***
C3 / DQS0_P	C3 / DQS0_P	***DQS***

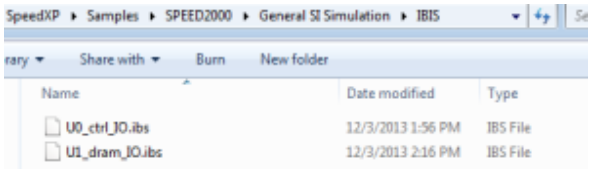
6. Check the newly generated models and click **OK** .
   The **Update IBIS File** dialog box opens to confirm changes.



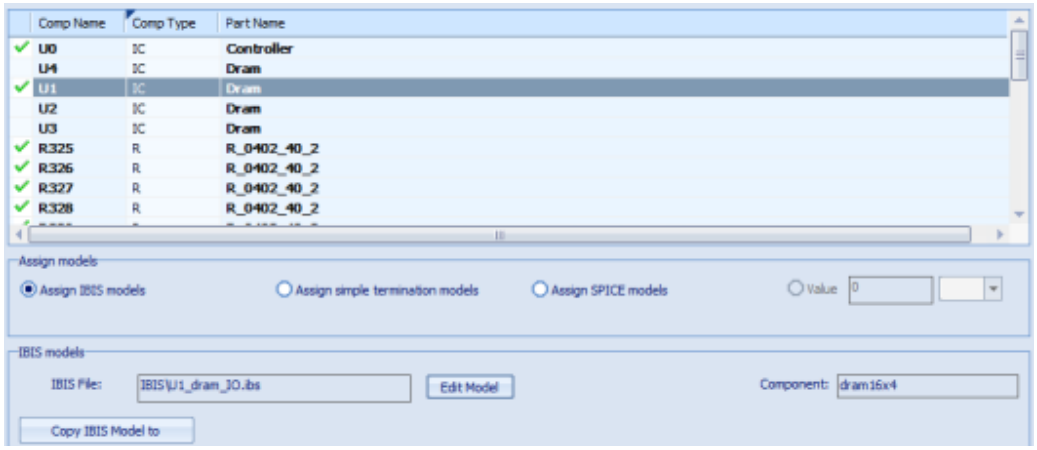
7. Click **Copy** .
   A dialog box opens to confirm the location where the IBIS file is copied to.



8. Click **OK** to create a new IBIS model.
   The **Assign IBIS models** window quits and a pin matched component IBIS model for controller **U1** is generated, located in the sub-folder named **IBIS** under the project folder.

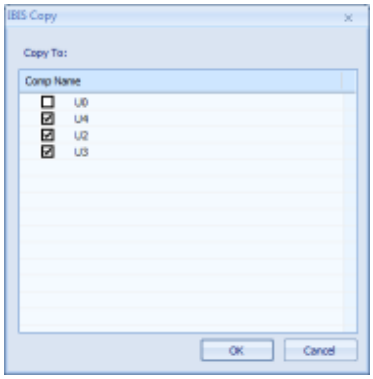


A green check mark appears ahead **U1** in the **Set up Component Models** window.

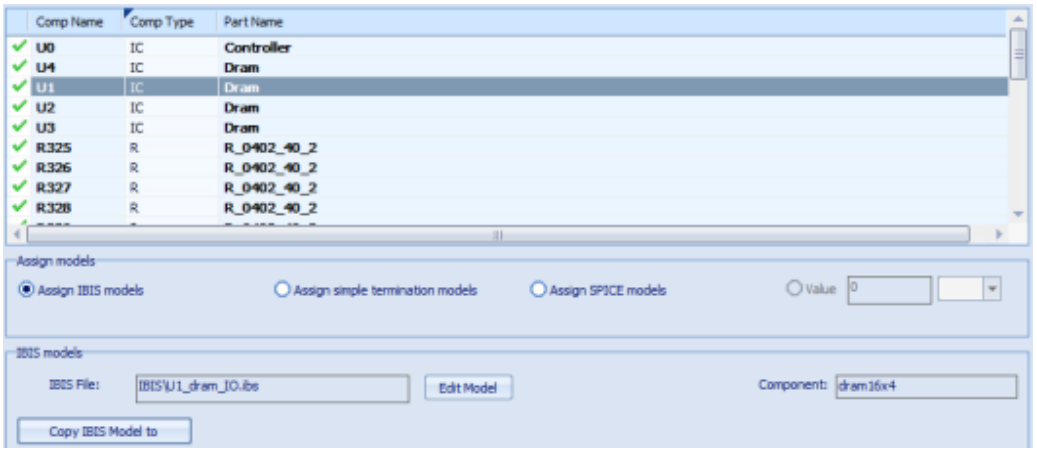


Copying IBIS Model to U2, U3 and U4 from U1

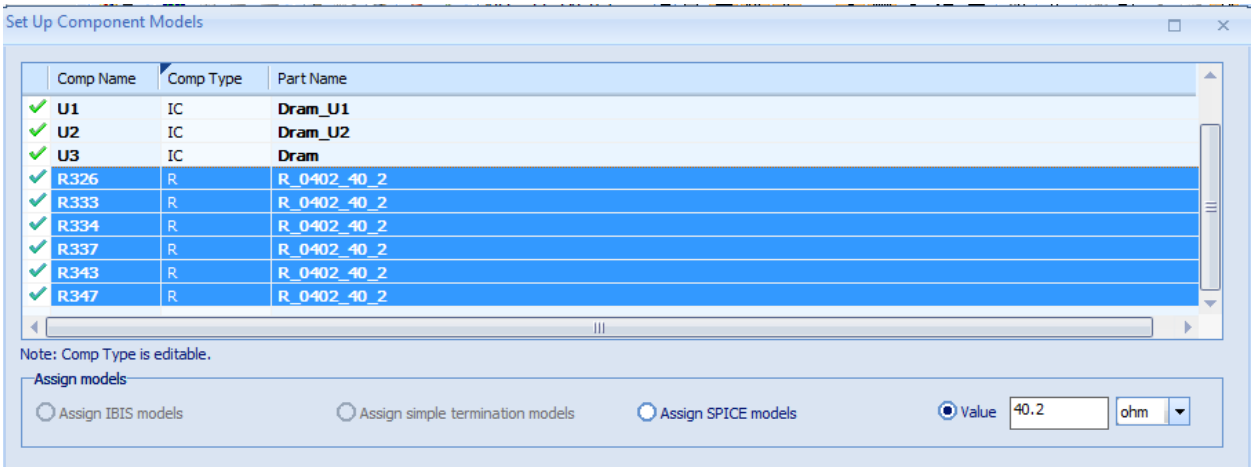
1. Click the **Copy IBIS Model to** button.
   The **IBIS Copy** window opens.
2. Select to check the components **U4** , **U2** and **U3** (uncheck **U0**).



3. Click **OK** .
   - All nets for component **U2** , **U3** and **U4** are assigned with IBIS models. .



   A green check mark appears ahead components **U4** , **U2** and **U3** in the **Set up Component Models** window. If the assigned IBIS models are not as desired, you can choose the corresponding component and click **Edit Model** button to change the IBIS model.
   - The value of passive component is generated automatically by default.



The value is editable (no change in this example).

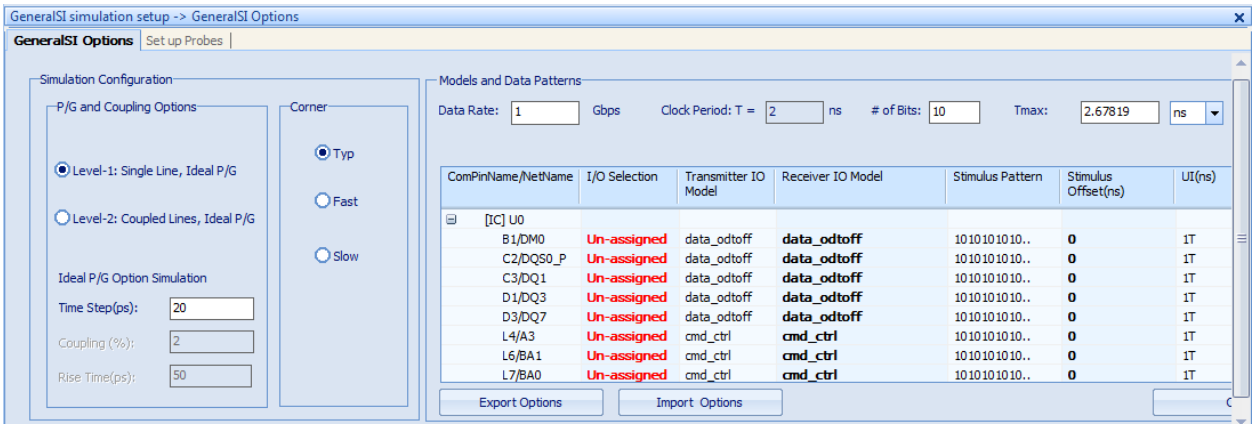
4. Click **OK** to exit the **Set up Component Models** window.

SI Simulation Options Setup

This section introduces how to set up SI simulation options.

Click **Set up SI Simulation Options** in the **Workflow** pane.

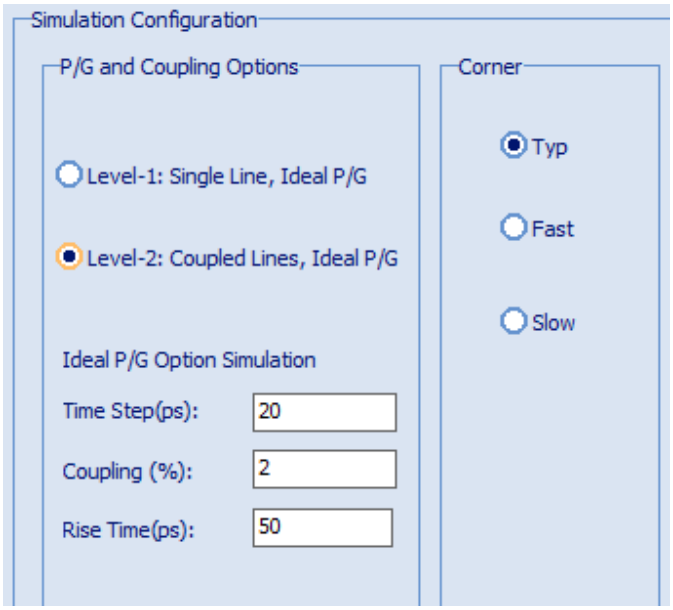
The **GeneralSI simulation setup -> GeneralSI Options** pane appears at the bottom of the window.



The pane contains two tabs: **GeneralSI Options** and **Set up Probes** .

General SI Options

1. Choose **Level-2: Coupled Lines, Ideal P/G** , and set the following parameters:
   - **Time Step(ps):** 20
   - **Coupling (%):** 2
   - **Rise Time(ps):** 50
   - **Corner:** Typ



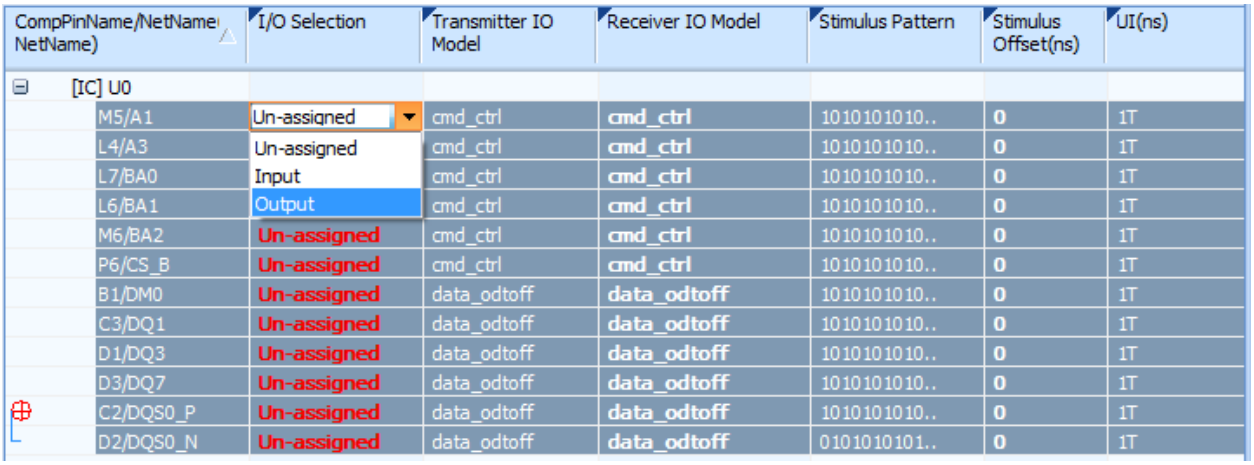
2. Input the following models and data parameters:
   - **Data Rate** : 1.333Gbps
   - **# of Bits** : 10
   - **Tmax** : 15ns

3. Click the column title **CompPinName/Netname** to reorder the nets alphabetically.

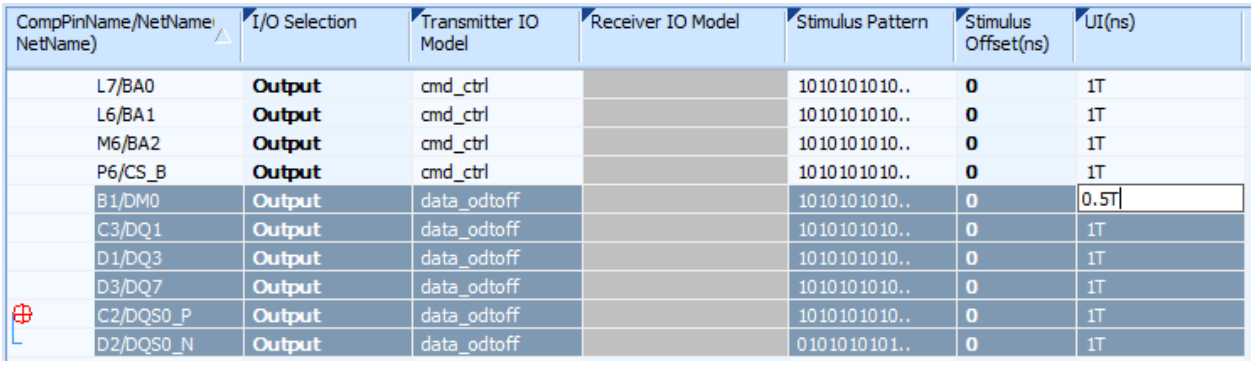
CompPinName/NetNa NetName)	I/O Selection	Transmitter IO Model	Receiver IO Model	Stimulus Pattern	Stimulus Offset(ns)	UI(ns)
[IC] U0						
M5/A1	Un-assigned	cmd_ctrl	**cmd_ctrl**	1010101010..	**0**	1T
L4/A3	Un-assigned	cmd_ctrl	**cmd_ctrl**	1010101010..	**0**	1T
L7/BA0	Un-assigned	cmd_ctrl	**cmd_ctrl**	1010101010..	**0**	1T
L6/BA1	Un-assigned	cmd_ctrl	**cmd_ctrl**	1010101010..	**0**	1T
M6/BA2	Un-assigned	cmd_ctrl	**cmd_ctrl**	1010101010..	**0**	1T
P6/CS_B	Un-assigned	cmd_ctrl	**cmd_ctrl**	1010101010..	**0**	1T
B1/DM0	Un-assigned	data_odtoff	**data_odtoff**	1010101010..	**0**	1T
C3/DQ1	Un-assigned	data_odtoff	**data_odtoff**	1010101010..	**0**	1T

U0 Parameters Setup

1. Choose to highlight all nets **A1** – **DQS0_0** under **U0** .
2. Click the icon in the field of **I/O Selection** column and select **Output** from the drop-down list.

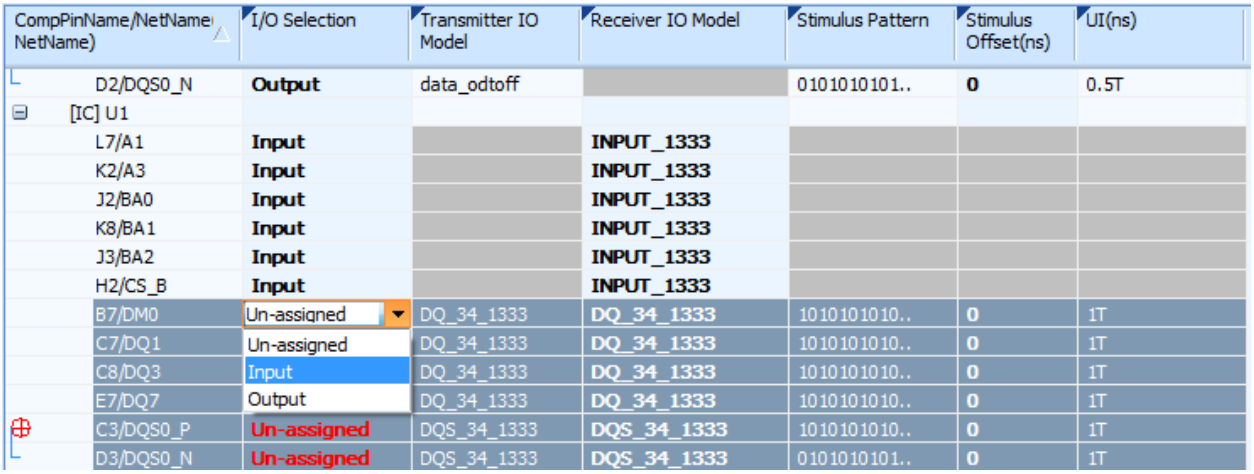


3. Select to highlight nets **DM0** – **DQS0_P** .
4. Click the icon in the field of **UI** column and input **0.5T** in the field.

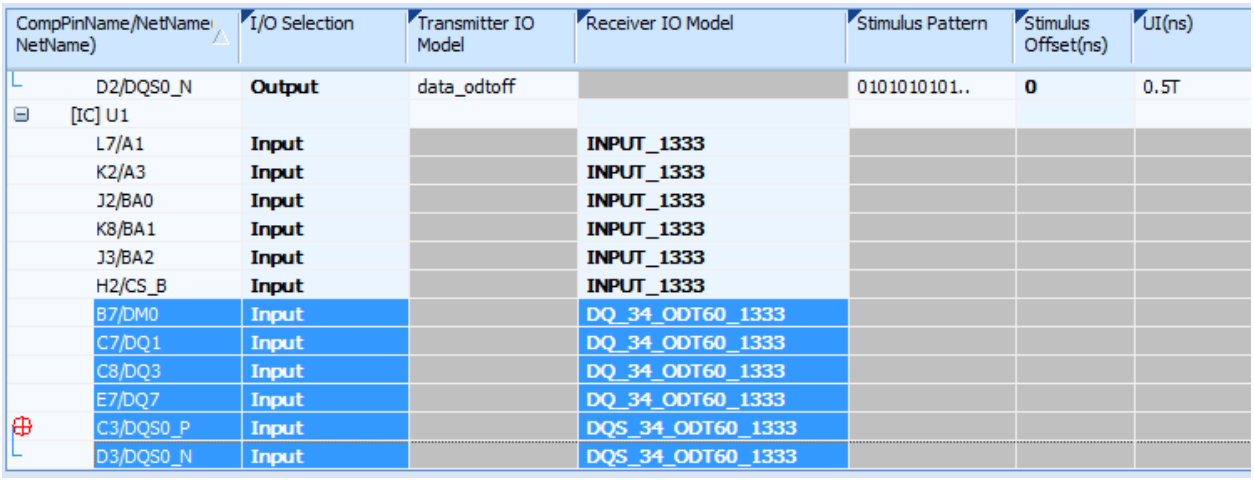


U1 Parameters Setup

1. Select to highlight nets **DM0 – DQS0_P** under **U1** .
2. Click the icon in the field of **I/O Selection** column and select **Input** from the drop-down list.



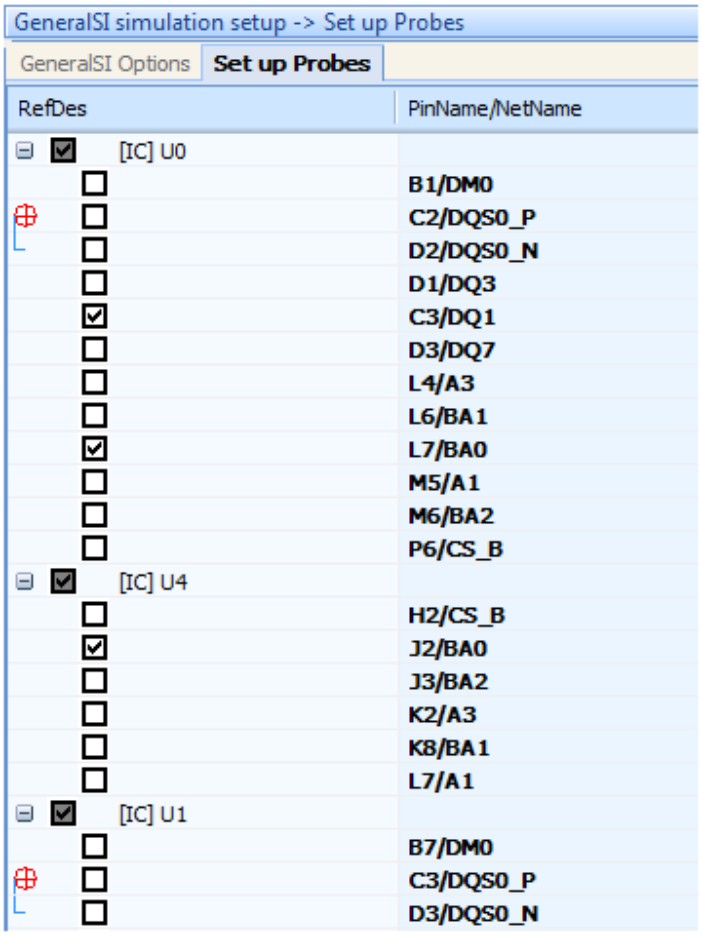
3. Assign the nets **DM0 – DQ7** with the **Receiver IO Model DQ_34_ODT60_1333** from the drop-down list.
4. Assign the nets **DQS0_P** and **DQS0_N** with **Receiver IO Model DQS_34_ODT60_1333** from the drop-down list.



5. Click **OK** to save all settings.

Probes Setup

1. Click **Set up Probes** in the **Workflow** pane.
   The **GeneralSI simulation setup -> Setup Probes** pane appears at the bottom of the window.

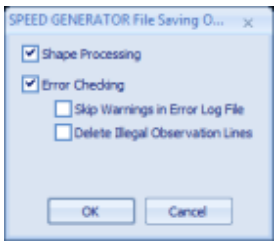


   You can also open the pane by clicking the **Setup Probes** tab in the above section.
   In this example, **BA0** and **DQ1** are primary nets. You can add the other probes of aggressor nets as desired.
2. Go back to the **GeneralSI simulation setup -> GeneralSI Options** pane.
3. Check all models and click **OK** to quit the pane.

Saving File

When all settings are complete, you can:

1. Click **Save File** in the **Workflow** pane to save the .spd files.
   A message window opens for you to confirm the selections.



2. Click **OK** with the default settings.

4

Simulation and Results

Click **Start Simulation** in the **Workflow** pane to perform simulation.

When the simulation is completed, the result waveforms are shown below.