

# **General\_SI\_Simulation\_Tutorial\_2**

**Product Version 16.6  
July 2014**

Legal

© 2014 Cadence Design Systems, Inc. All rights reserved.  
Printed in the United States of America.

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

Trademarks: Trademarks and service marks of Cadence Design Systems, Inc. (Cadence) contained in this document are attributed to Cadence with the appropriate symbol. For queries regarding Cadence's trademarks, contact the corporate legal department at the address shown above or call 1-800-862-4522.

All other trademarks are the property of their respective holders.

Restricted Print Permission: This publication is protected by copyright and any unauthorized use of this publication may violate copyright, trademark, and other laws. Except as specified in this permission statement, this publication may not be copied, reproduced, modified, published, uploaded, posted, transmitted, or distributed in any way, without prior written permission from Cadence. This statement grants you permission to print one (1) hard copy of this publication subject to the following conditions:

1. The publication may be used solely for personal, informational, and noncommercial purposes;
2. The publication may not be modified in any way;
3. Any copy of the publication or portion thereof must include all original copyright, trademark, and other proprietary notices and this permission statement; and
4. Cadence reserves the right to revoke this authorization at any time, and any such use shall be discontinued immediately upon written notice from Cadence.

Disclaimer: Information in this publication is subject to change without notice and does not represent a commitment on the part of Cadence. The information contained herein is the proprietary and confidential information of Cadence or its licensors, and is supplied subject to, and may be used only by Cadence's customer in accordance with, a written agreement between Cadence and its customer. Except as may be explicitly set forth in such agreement, Cadence does not make, and expressly disclaims, any representations or warranties as to the completeness, accuracy or usefulness of the information contained in this document. Cadence does not warrant that use of such information will not infringe any third party rights, nor does Cadence assume any liability for damages or costs of any kind that may result from use of such information.

Restricted Rights: Use, duplication, or disclosure by the Government is subject to restrictions as set forth in FAR52.227-14 and DFAR252.227-7013 et seq. or its successor.

---

# 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

---

# Introduction

---

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

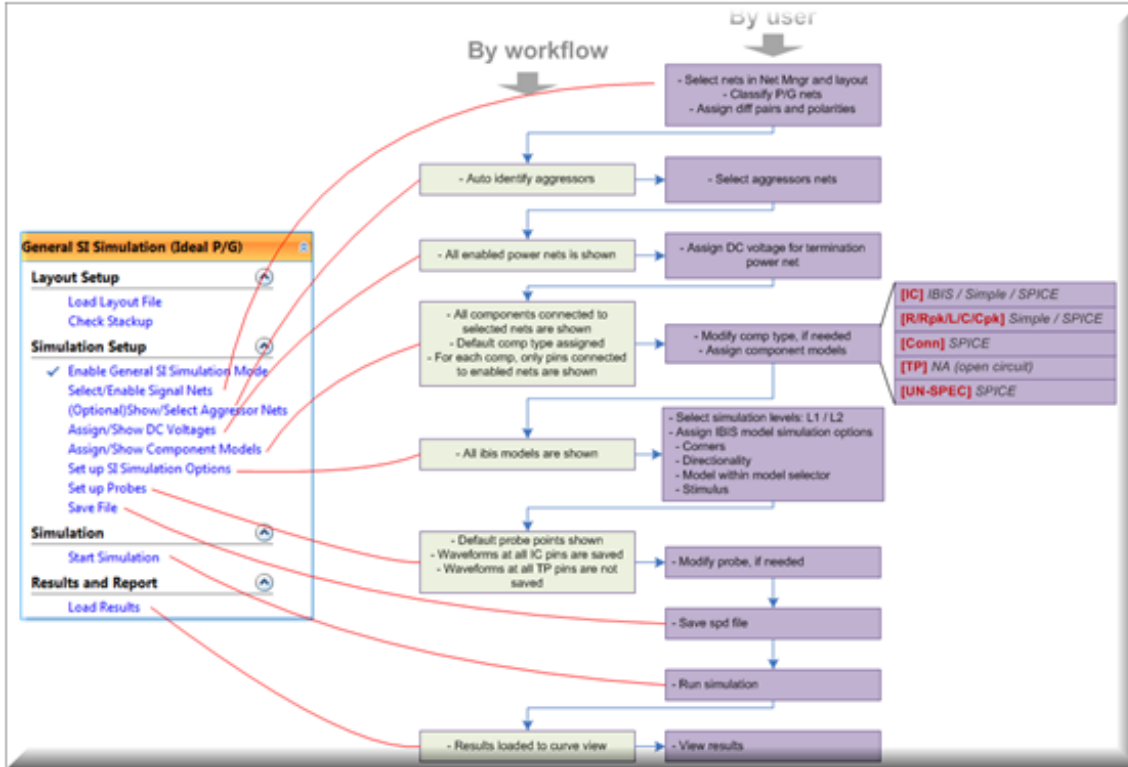
## Overview

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

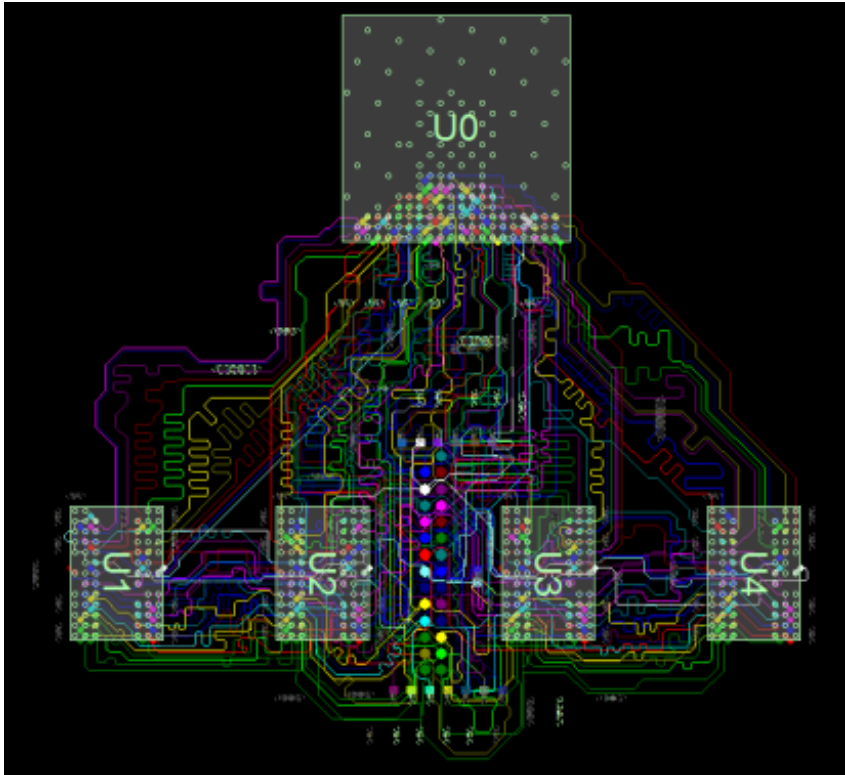
In General SI Simulation (Ideal/P/G) workflow, you can firstly enable signal nets to be simulated, which are referred as primary nets. In order to better simulate xtalk effects, a new feature - to select which aggressor nets to be included in simulation (optional) - is added to find all the aggressor 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.

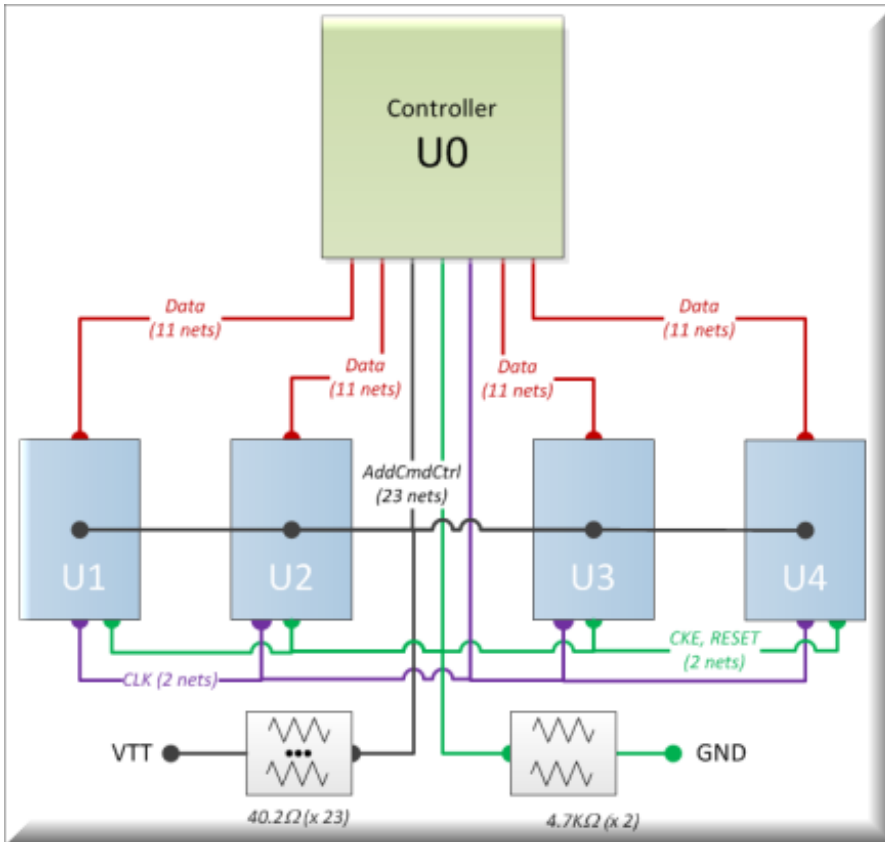


## Tutorial Summary

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

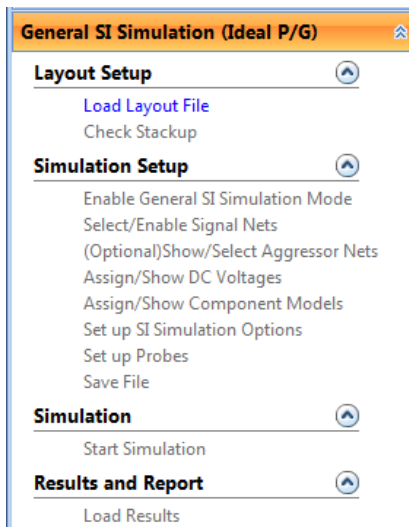
- Run simulation
- View simulation results



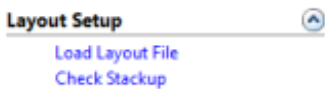
# Layout Setup

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

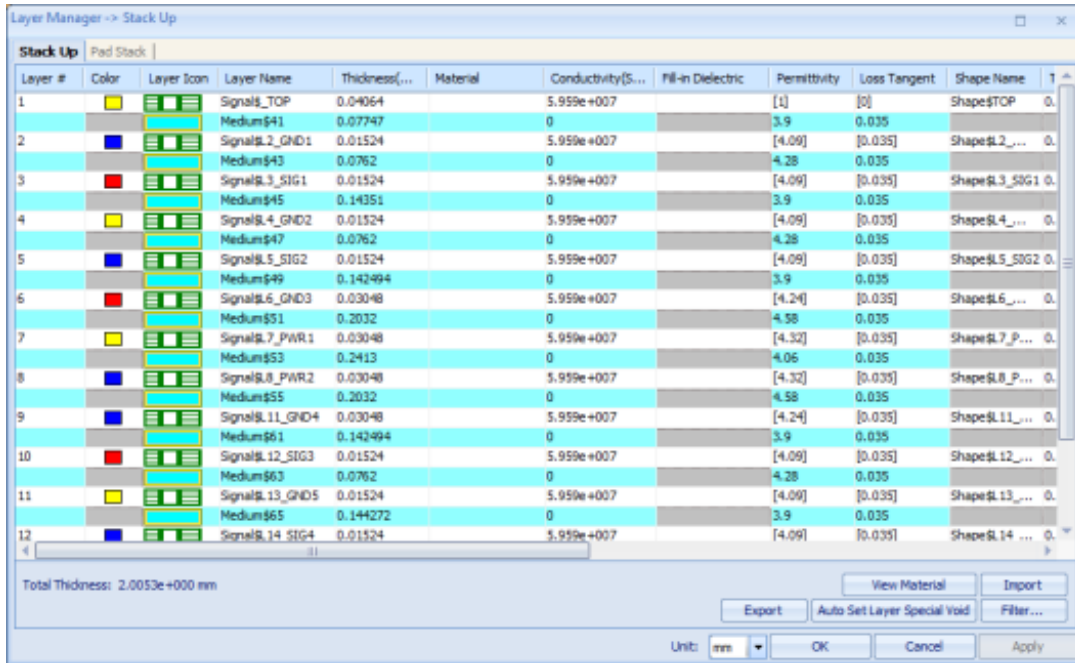
1. Launch **SPEED2000 Generator** , and choose the **General SI Simulation (Ideal P/G)** workflow.



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.

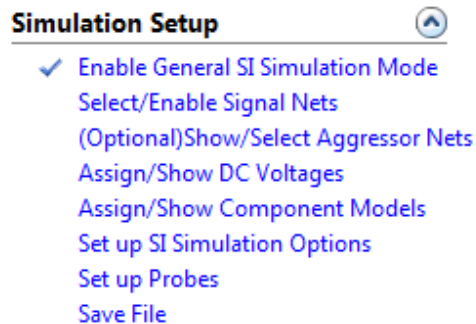
---

## 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.

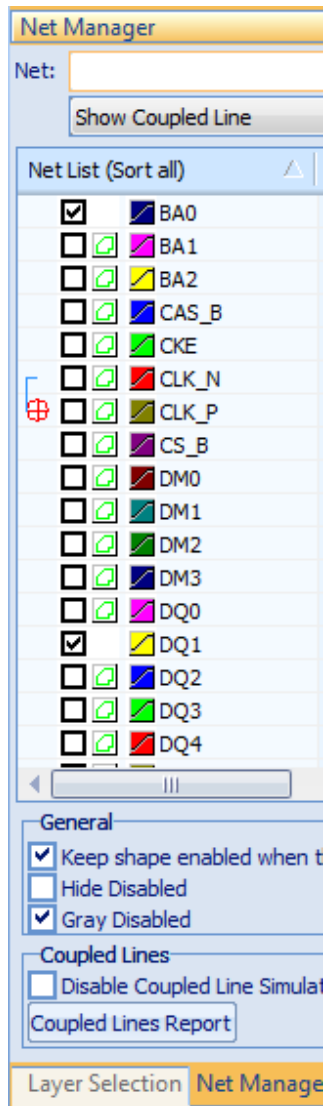


When enabled, a check mark ✓ appears ahead the workflow step. And all other related steps are enabled.

## 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.

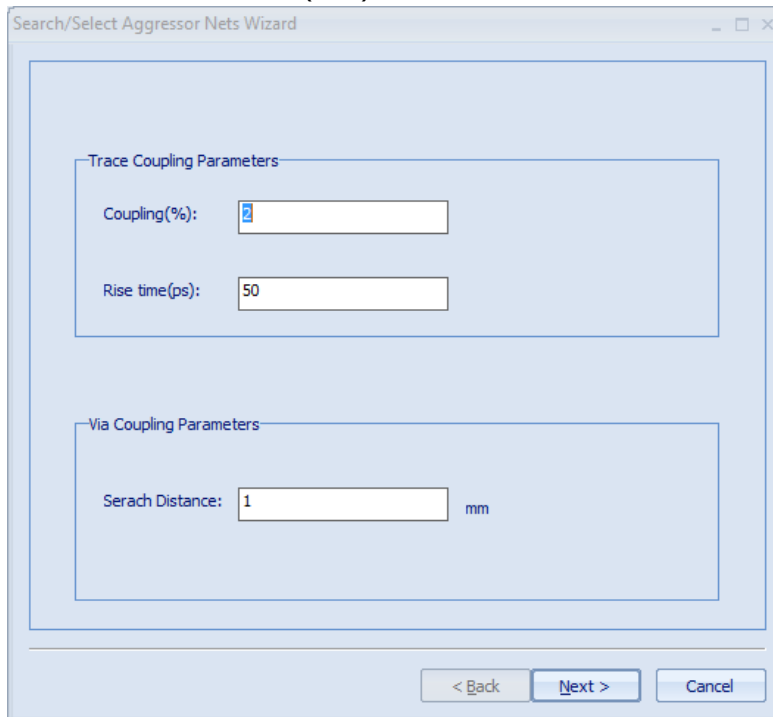
- If the diff-pair is not guessed out, choose both positive and negative nets, right-click and choose **Classify as diff pair** to define it manually.
- If the polarity is not guessed out, right-click the positive net and choose **Switch Polarity** to define it manually.

## Showing Aggressor Nets

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

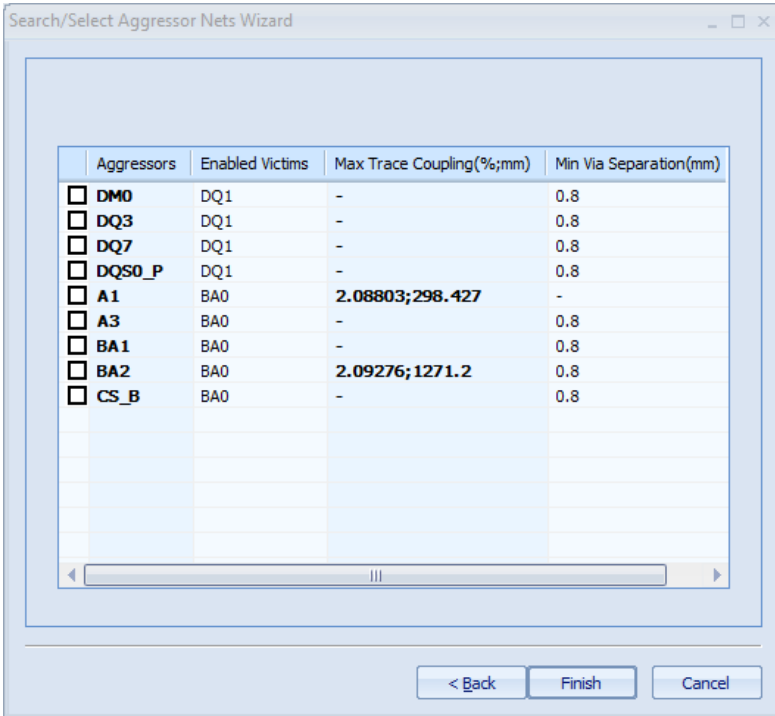
coupling. You can manually choose the desired aggressor signal nets to be included in simulation.

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)



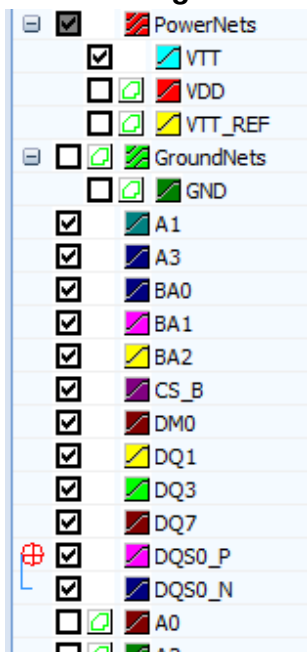
The screenshot shows a dialog box titled "Search/Select Aggressor Nets Wizard". It has a light blue background and a white border. The dialog is divided into two main sections. The first section, "Trace Coupling Parameters", contains two input fields: "Coupling(%)" with the value "2" and "Rise time(ps)" with the value "50". The second section, "Via Coupling Parameters", contains one input field: "Search Distance" with the value "1" and the unit "mm". At the bottom of the dialog, there are three buttons: "< Back", "Next >", and "Cancel".

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 Aggressor Nets Wizard** displays all aggressors nets.



4. Click to enable all nets in this sample and click **Finish** .

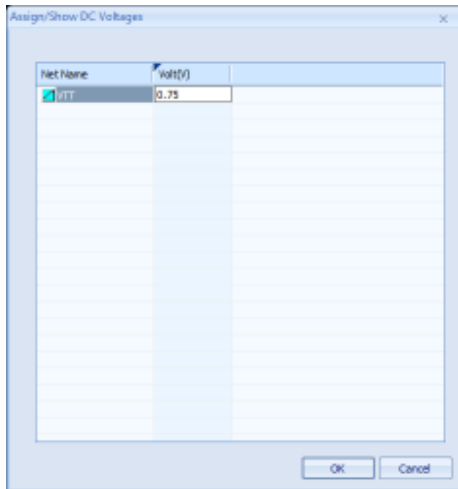
The **Search/Select Aggressor Nets Wizard** exists and the corresponding nets are enabled in **Net Manager** .



## 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.



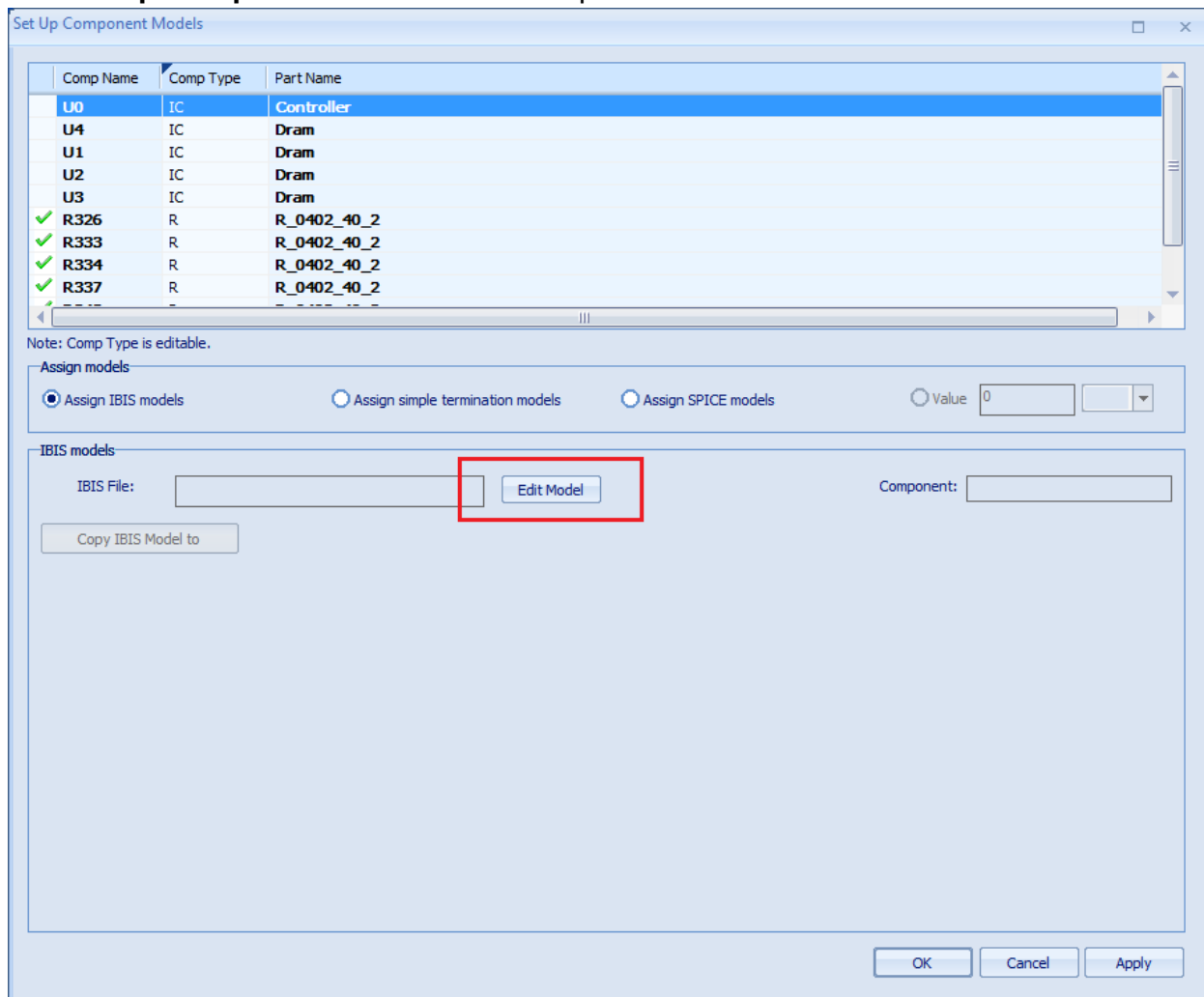
3. Click **OK** .

## Assigning Component Models

This section introduces how to assign models to all components.

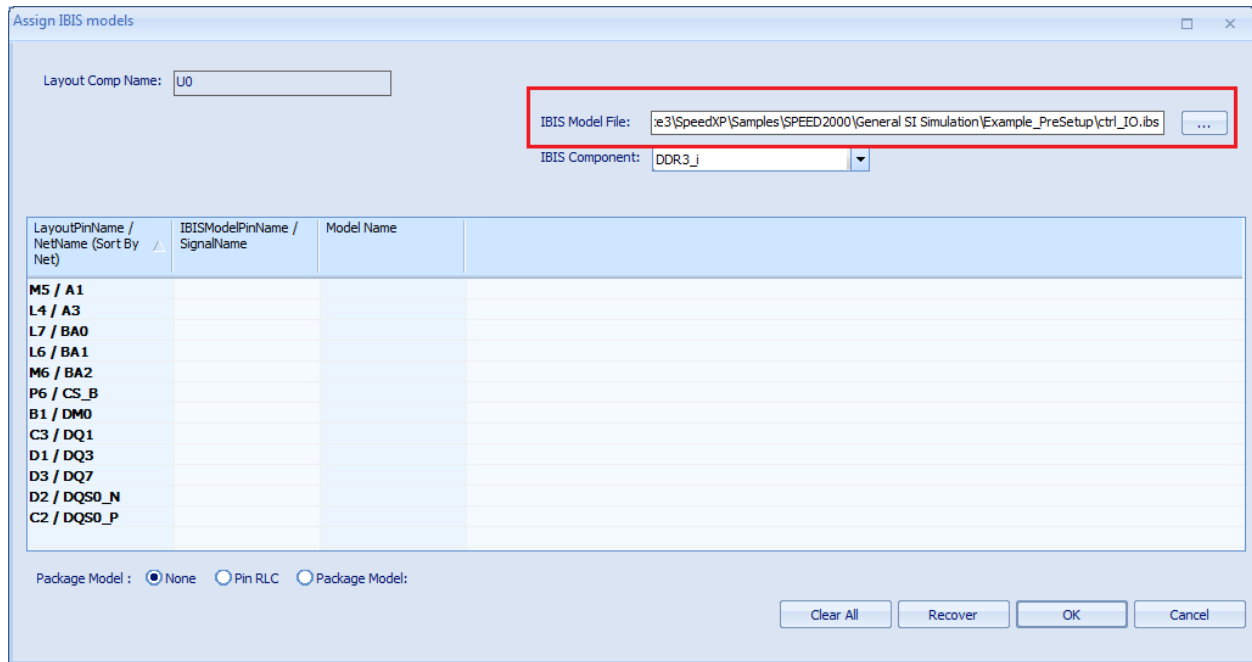
## Assigning IBIS Model to IC Component U0


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

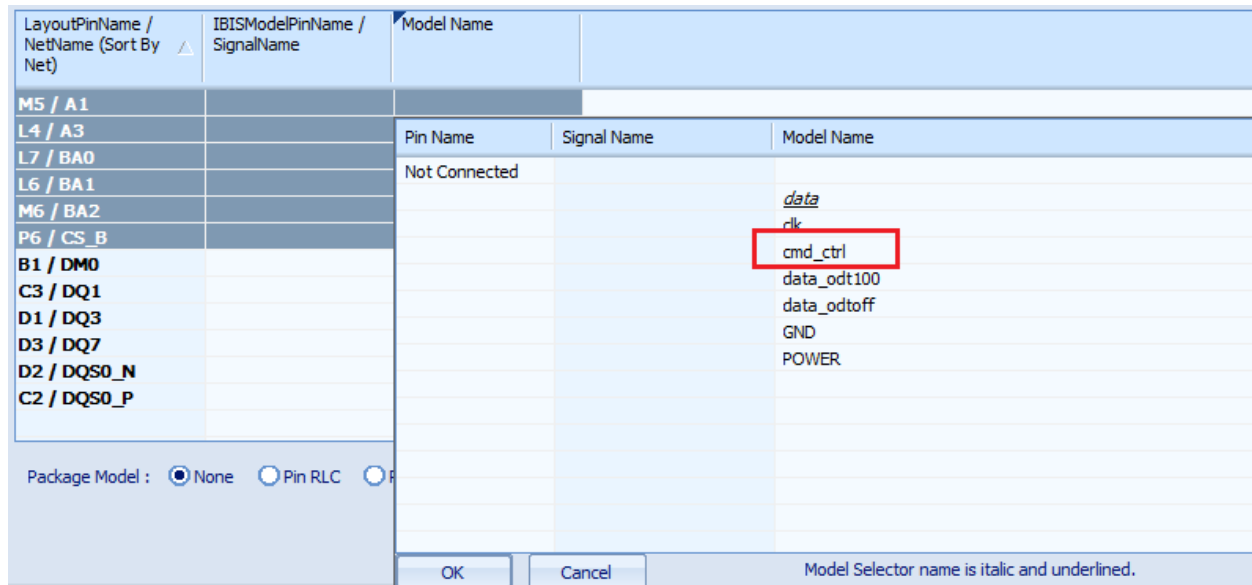


2. Select to highlight component **U0** and click the **Edit Model** button.  
The **Assign IBIS models** window opens.





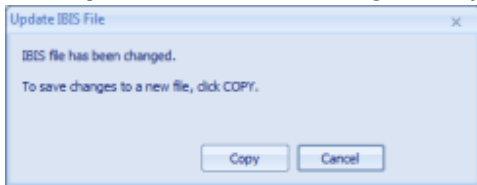
- Browse to load the IBIS file ctrl\_IO.ibs.  
The IBIS file is loaded.
- Use **Shift** and **Ctrl** keys to select multiple nets **A1 – CS\_B**.
- Click the icon  in the field of **Model Name** column and select **cmd\_ctrl** from the pop-up window.



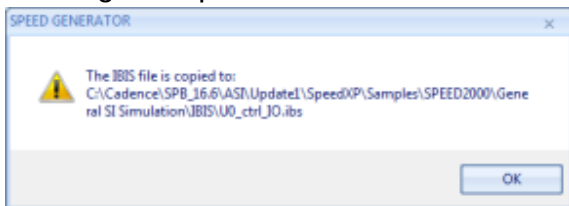
- Click **OK**.  
The selected nets are assigned with the model **cmd\_ctrl**.  
Repeat the above steps to assign nets **DM0 – DQS0\_P** with the model **data**.  
All nets are assigned with IBIS models.

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	<i>data</i>
C3 / DQ1	C3 / DQ1	<i>data</i>
D1 / DQ3	D1 / DQ3	<i>data</i>
D3 / DQ7	D3 / DQ7	<i>data</i>
D2 / DQS0_N	D2 / DQS0_N	<i>data</i>
C2 / DQS0_P	C2 / DQS0_P	<i>data</i>

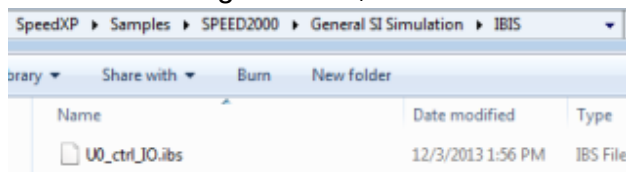
7. Check the newly generated models and click **OK** .  
The **Update IBIS File** dialog box opens to confirm the changes.



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

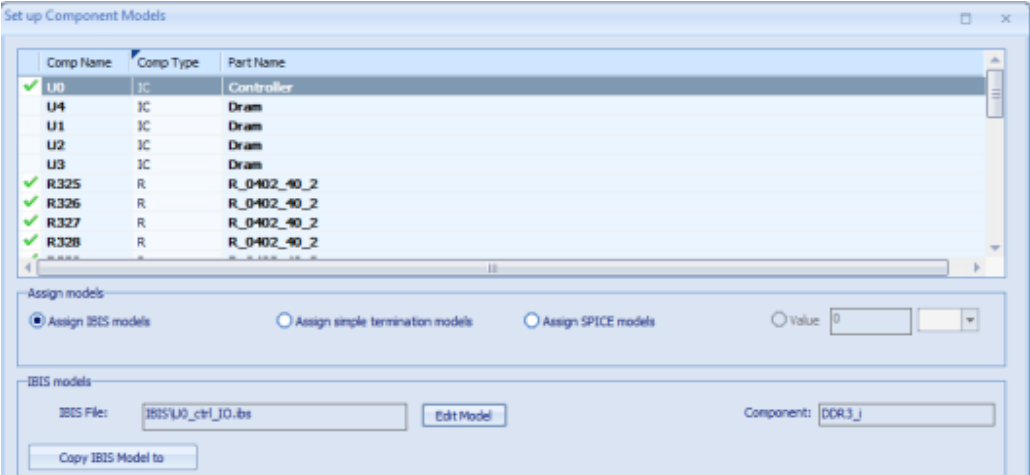


9. Click **OK** to create a new IBIS model.  
The **Assign IBIS models** window quits and a pin matched component IBIS model for controller **U0** is generated, located in the sub-folder named **IBIS** under the project folder.



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

General\_SI\_Simulation\_Tutorial\_2  
Simulation Setup

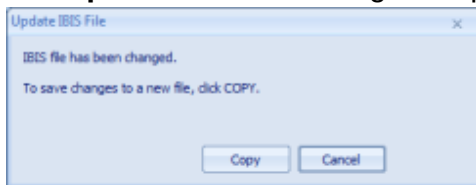


## Assigning IBIS Model to IC Component U1

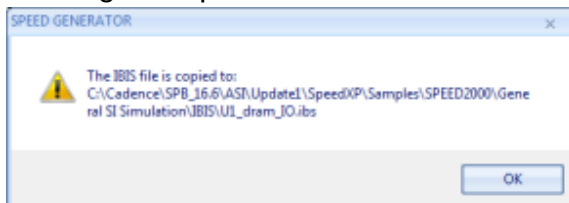
1. Select to highlight component **U1** and click the **Edit Model** button.  
The **Assign IBIS models** window opens.
  2. Browse to load the IBIS file dram\_IO.ibs.  
The IBIS file is loaded.
  3. Repeat the steps described in the above section to assign nets **A1 – CS\_B** with the model **INPUT**.
  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	<b>INPUT</b>
K2 / A3	K2 / A3	<b>INPUT</b>
J2 / BA0	J2 / BA0	<b>INPUT</b>
K8 / BA1	K8 / BA1	<b>INPUT</b>
J3 / BA2	J3 / BA2	<b>INPUT</b>
H2 / CS_B	H2 / CS_B	<b>INPUT</b>
B7 / DM0	B7 / DM0	<b>DQ</b>
C7 / DQ1	C7 / DQ1	<b>DQ</b>
C8 / DQ3	C8 / DQ3	<b>DQ</b>
E7 / DQ7	E7 / DQ7	<b>DQ</b>
D3 / DQS0_N	D3 / DQS0_N	<b>DQS</b>
C3 / DQS0_P	C3 / DQS0_P	<b>DQS</b>

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

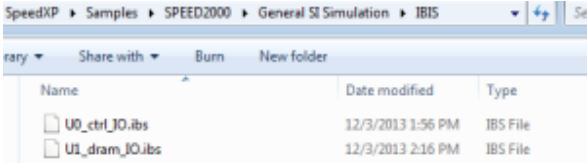


7. Click **Copy**.  
A dialog box opens to confirm the location where the IBIS file is copied to.

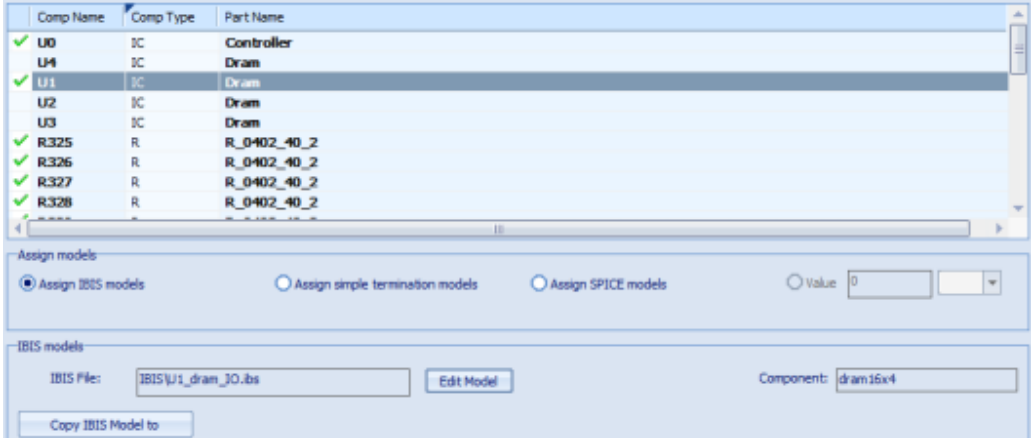


8. Click **OK** to create a new IBIS model.  
The **Assign IBIS models** window quits and a pin matched component IBIS model for controller **U1** is generated, located in the sub-folder named **IBIS** under the project folder.

General\_SI\_Simulation\_Tutorial\_2  
Simulation Setup

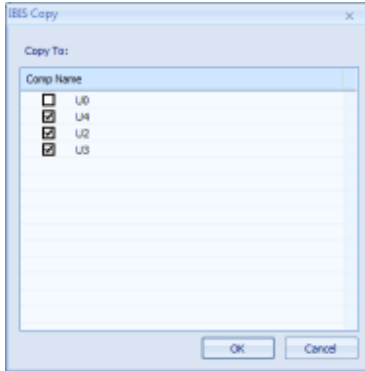


A green check mark appears ahead **U1** in the **Set up Component Models** window.

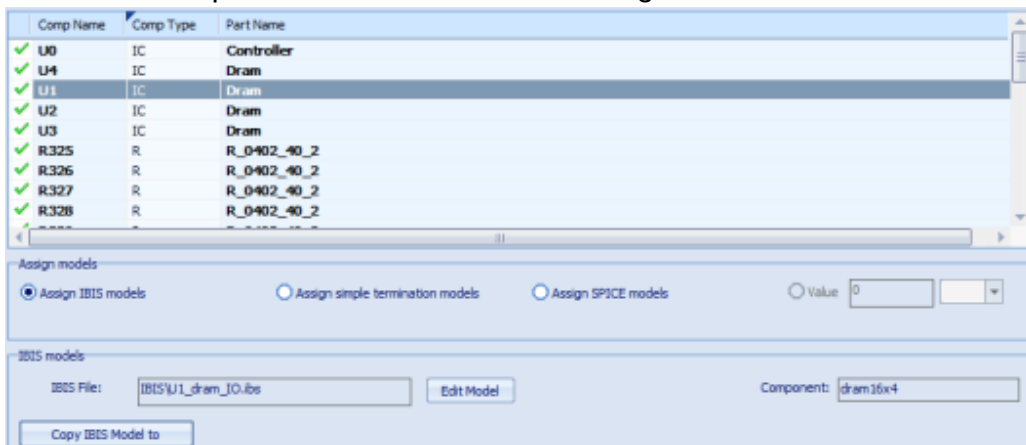


## Copying IBIS Model to U2, U3 and U4 from U1

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** ).

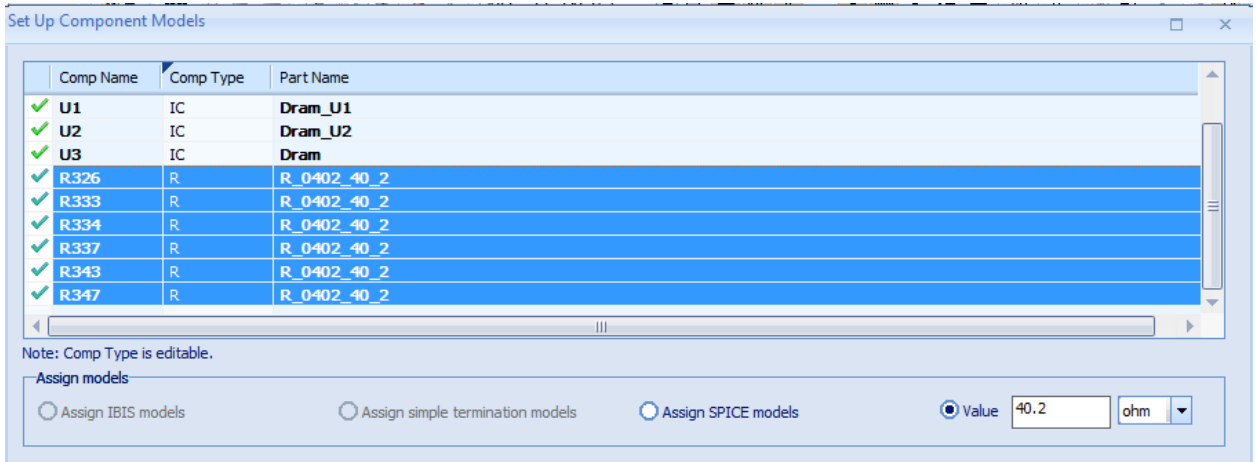


3. Click **OK** .
  - All nets for component **U2** , **U3** and **U4** are assigned with IBIS models. .



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).

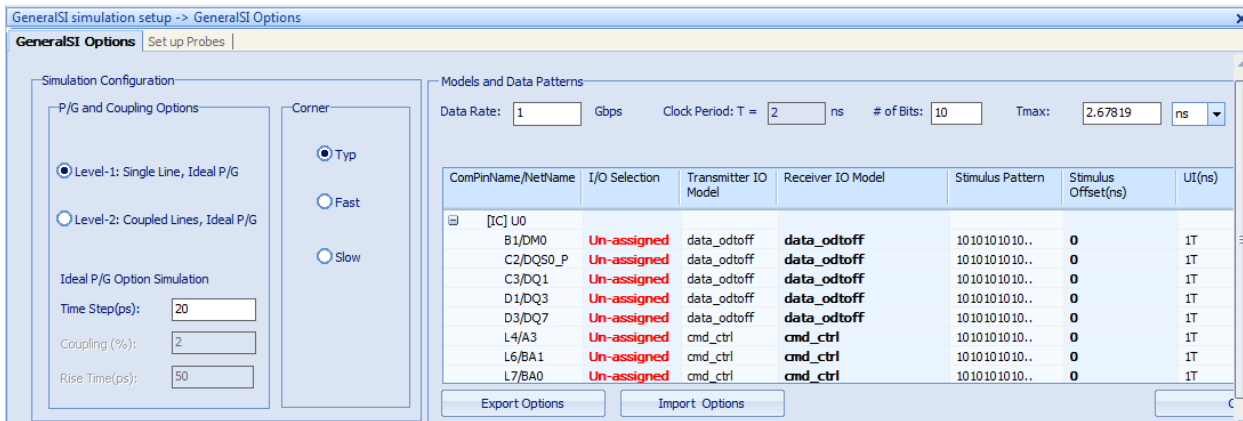
4. Click **OK** to exit the **Set up Component Models** window.

## 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.

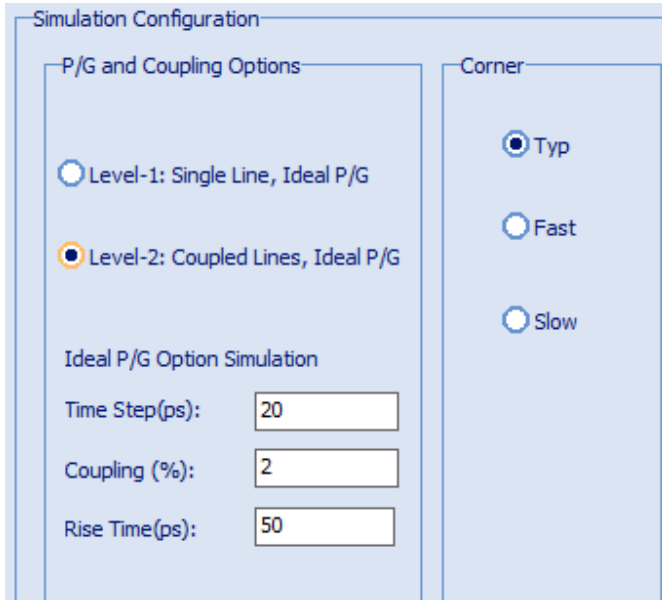


The pane contains two tabs: **GeneralSI Options** and **Set up Probes**.

## General SI Options

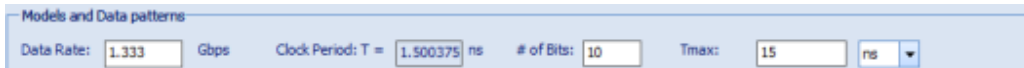
1. Choose **Level-2: Coupled Lines, Ideal P/G** , and set the following parameters:

- **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/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.

CompPinName/NetName/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	Input	cmd_ctrl	cmd_ctrl	1010101010..	0	1T
L6/BA1	Output	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
D1/DQ3	Un-assigned	data_odtoff	data_odtoff	1010101010..	0	1T
D3/DQ7	Un-assigned	data_odtoff	data_odtoff	1010101010..	0	1T
C2/DQS0_P	Un-assigned	data_odtoff	data_odtoff	1010101010..	0	1T
D2/DQS0_N	Un-assigned	data_odtoff	data_odtoff	0101010101..	0	1T

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.

CompPinName/NetName/NetName)	I/O Selection	Transmitter IO Model	Receiver IO Model	Stimulus Pattern	Stimulus Offset(ns)	UI(ns)
L7/BA0	Output	cmd_ctrl		1010101010..	0	1T
L6/BA1	Output	cmd_ctrl		1010101010..	0	1T
M6/BA2	Output	cmd_ctrl		1010101010..	0	1T
P6/CS_B	Output	cmd_ctrl		1010101010..	0	1T
B1/DM0	Output	data_odtoff		1010101010..	0	0.5T
C3/DQ1	Output	data_odtoff		1010101010..	0	1T
D1/DQ3	Output	data_odtoff		1010101010..	0	1T
D3/DQ7	Output	data_odtoff		1010101010..	0	1T
C2/DQS0_P	Output	data_odtoff		1010101010..	0	1T
D2/DQS0_N	Output	data_odtoff		0101010101..	0	1T

## 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.

CompPinName/NetName/ NetName)	I/O Selection	Transmitter IO Model	Receiver IO Model	Stimulus Pattern	Stimulus Offset(ns)	UI(ns)
D2/DQS0_N	Output	data_odtoff		0101010101..	0	0.5T
[IC] U1						
L7/A1	Input		INPUT_1333			
K2/A3	Input		INPUT_1333			
J2/BA0	Input		INPUT_1333			
K8/BA1	Input		INPUT_1333			
J3/BA2	Input		INPUT_1333			
H2/CS_B	Input		INPUT_1333			
B7/DM0	Un-assigned	DQ_34_1333	DQ_34_1333	1010101010..	0	1T
C7/DQ1	Un-assigned	DQ_34_1333	DQ_34_1333	1010101010..	0	1T
C8/DQ3	Input	DQ_34_1333	DQ_34_1333	1010101010..	0	1T
E7/DQ7	Output	DQ_34_1333	DQ_34_1333	1010101010..	0	1T
C3/DQS0_P	Un-assigned	DQS_34_1333	DQS_34_1333	1010101010..	0	1T
D3/DQS0_N	Un-assigned	DQS_34_1333	DQS_34_1333	0101010101..	0	1T

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.

CompPinName/NetName/ NetName)	I/O Selection	Transmitter IO Model	Receiver IO Model	Stimulus Pattern	Stimulus Offset(ns)	UI(ns)
D2/DQS0_N	Output	data_odtoff		0101010101..	0	0.5T
[IC] U1						
L7/A1	Input		INPUT_1333			
K2/A3	Input		INPUT_1333			
J2/BA0	Input		INPUT_1333			
K8/BA1	Input		INPUT_1333			
J3/BA2	Input		INPUT_1333			
H2/CS_B	Input		INPUT_1333			
B7/DM0	Input		DQ_34_ODT60_1333			
C7/DQ1	Input		DQ_34_ODT60_1333			
C8/DQ3	Input		DQ_34_ODT60_1333			
E7/DQ7	Input		DQ_34_ODT60_1333			
C3/DQS0_P	Input		DQS_34_ODT60_1333			
D3/DQS0_N	Input		DQS_34_ODT60_1333			

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.

RefDes	PinName/NetName
<input checked="" type="checkbox"/> [IC] U0	
<input type="checkbox"/>	B1/DM0
<input type="checkbox"/>	C2/DQS0_P
<input type="checkbox"/>	D2/DQS0_N
<input type="checkbox"/>	D1/DQ3
<input checked="" type="checkbox"/>	C3/DQ1
<input type="checkbox"/>	D3/DQ7
<input type="checkbox"/>	L4/A3
<input type="checkbox"/>	L6/BA1
<input checked="" type="checkbox"/>	L7/BA0
<input type="checkbox"/>	M5/A1
<input type="checkbox"/>	M6/BA2
<input type="checkbox"/>	P6/CS_B
<input checked="" type="checkbox"/> [IC] U4	
<input type="checkbox"/>	H2/CS_B
<input checked="" type="checkbox"/>	J2/BA0
<input type="checkbox"/>	J3/BA2
<input type="checkbox"/>	K2/A3
<input type="checkbox"/>	K8/BA1
<input type="checkbox"/>	L7/A1
<input checked="" type="checkbox"/> [IC] U1	
<input type="checkbox"/>	B7/DM0
<input type="checkbox"/>	C3/DQS0_P
<input type="checkbox"/>	D3/DQS0_N

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.

# Simulation and Results

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

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

