DDR4 Simulation Guidelines

Introduction

These guidelines were created for use with the existing Memory Design Guidelines and provide information on the simulations necessary to create the information needed for those guidelines that are very important to. This set of guidelines will be created using Arria 10 IBIS models.

Background Knowledge Source

The Altera External Memory Interface Handbook provides a thorough explanation of DDR4 topologies and board design guidelines for DDR4 systems. The External Memory Interface (EMIF) Handbook is very useful in understanding what needs to be done to create a successful system.

http://www.altera.com/literature/hb/external-memory/emi_plan.pdf

The EMIF handbook guidelines were created using useful numbers for spacing of the traces and other constraints for the system. If a 2D simulator available figures can be created for signal parallelism rules and length matching. Also duty cycle distortion for clocks and DQS signals can be examined to avoid timing problems in systems with high data rates.

In the document below the user will be lead through the simulation process using the 2D HyperLynx to determine the optimum drive and termination levels for the pseudo open drain (POD) DQ/DQS interface and create informed parallelism constraints for the topology.

Contents

Chapter 1: Background

Properties of DDR4 systems that have an effect on simulation

The following table of properties constrains what needs to be simulated. Your requirements will vary due to device speeds and timing constraints.

Table 1

Property	Value	Notes
I/O Voltage	1.2V	L version could be 1.05
Data Rates	1600MHz to 3200Mbps. [Arria 10 doesn't support full range. Current supported range is 1333 MHz max]	800MHz to 1.6GHz Clock (See note 3)
Clock Signals	SSTL 1.2V differential	Externally Terminated (See note 1)
Address and Control Signal Standard	SSTL 1.2V	Externally Terminated (See note 1)
Address and Control Signal Rate	1T or 2T	(See note 4)
DQ bus	POD12	Lower SSN for systems
POD output drive strength	40 Ohms	Altera Standard (See note 2)
Board trace characteristic impedance	40 Ohms	Strongly Recommended (Micron)

Notes for Table 1:

1. The Address, Command and Clock signals all use a threshold of 0.49*VCC. For data rates up to 2133MTs the thresholds are +/-125mv. For 2400MTs the thresholds are +/-100mv.
2. POD12 is the acronym for the Pseudo Open Drain 1.2V interface. This interface is terminated to VCC(1.2V) to reduce simultaneous signaling noise and reduce the complexity of the system. The threshold voltage is therefore not 1/2VCC but something much higher. This voltage also depends on the strength of the driver in the long term so DDR4 memory devices have an adaptable threshold built in and bus inversion is included to help minimize DC drift in the system. The DDR4 JEDEC specification for drive strength is 39 Ohms.
3. The specification for DDR4 gives a clock range of 1.25ns maximum to 0.625ns minimum or 800MHz to 1600MHz.
4. The data rate is equal to the clock rate for 1T and half the clock rate for 2T. Using the 2T timing allows much more time for the signals to stabilize.

General Simulation Method

In order to assure that a memory project will be successful a set of simulations should be created. This document covers the simulation methods necessary for DDR4.

The following subjects will be covered.

- Necessary Input
    - Topology
    - Data Rate
    - Stackup
    - Models
- Pre-layout Simulations
    - Command/Address group
    - Clock signals
    - DQ/DQS Groups
- Post-layout Simulations

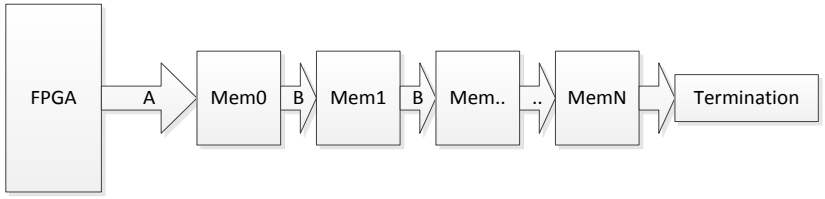
Topology

DDR4 was designed to use a Fly-by topology for the Command/Address and Clock system. The DQ/DQS systems use read and write leveling to provide accurate timing for the exchange of data. The simulation set necessary for each group is unique.

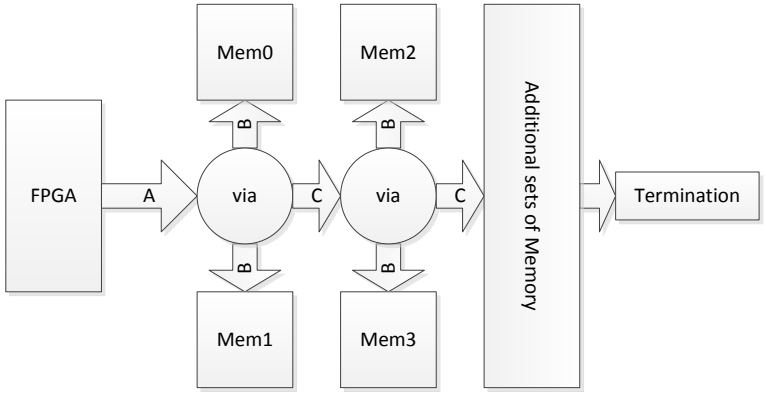
The following topology details are necessary.

- The number of devices in the chain
- The expected distance between the FPGA, the memory devices, and the termination

Fly-by with equidistant memories



Fly-by with "Ping Pong" configuration



The challenge with the "Ping Pong" arrangement is lower impedance due to the capacitance of closely associated memory parts and branching of the controlled impedance paths.

Pre-Layout Simulations

Pre-layout simulations provide information on what is needed to create a successful topology. A schematic is created using symbols for all of components and connections in the desired topology. Simulations are executed to provide information on the expected channel response and a graphical display is shown to use in analyzing the signal at the source and destination(s).

Chapter 2: Board data and Simulation Models

Stackup

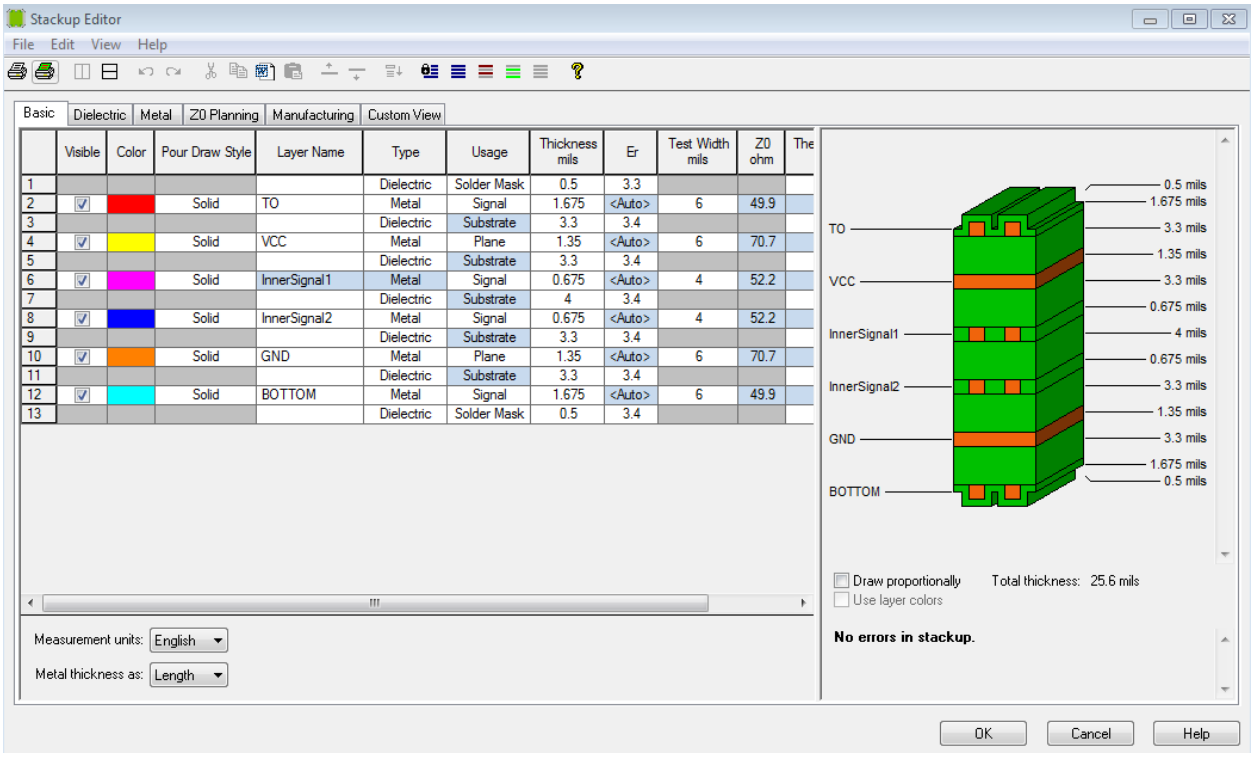
In pre-layout simulations it is best to use the stackup that will be used for the PCB. Sometimes the final stackup is not available and one a simulation stackup can be created using the dielectric constant (Er) and the dielectric loss tangent (tanδ) of what will be the target dielectric. The Er only determines the propagation delay so whatever its value only changes the timing of the wave. The tanδ is important because it absorbs the high frequency portions of the waveform. This loss removes harmonics, causing smoothing of the corners of the waveforms and decreasing the risetime, therefor delaying the time of flight. A higher loss dielectric can be used to reduce the edge speed of a signal and reduce reflections as long as it does not interfere with transceiver signals that may accidently be placed on the routing layers for memory signals.

It is critical that, for microstrip simulations, the characteristics of the soldermask be known. Usually soldermask materials have a really high tanδ that make life interesting for fast edges.

Stackup setup in HyperLynx

A stackup is needed to establish a baseline for the project. For this project the stackup data for the Arria 10 FPGA development kit (PCIe) was copied. Megrton6 was used for the board because of the transceiver requirements. Be sure to use the material that will be used to implement your project. Dielectric values are readily available from PCB manufacturers as well as from dielectric vendors.

If you have significant high-speed signal runs on the microstrip layer then be sure to have accurate data on the soldermask layer.



After the circuit board layout is created, it can be imported into HyperLynx and the stackup will come with it. It is therefore important to be sure that the stackup for the board has accurate data on the actual board parameters for Dk and Df. (Er and Tanδ).

Simulation Models

For pre-layout simulations there are two options for obtaining IBIS files for use with HyperLynx. Models can be obtained from the Altera website or created from a Quartus® II project. Two types of models can be used, depending on where you are in the design of the FPGA.

The first is a set of IBIS models for all the IO capability available for the FPGA. These models have a generic set of values for the package RLC values.

The second is a set of IBIS models with models specific to the implementation and accurate RLC values for each pin. This file is very much smaller than the one you will get from the website.

Altera IBIS models from the website

An IBIS file with models of all the possible I/O standards and strengths can be found by following the link below. There are multiple files for Arria 10 but the one that will have the models for DDR4 is Arria10.zip.

https://www.altera.com/support/support-resources/download/board-layout-test/ibis/ibs-ibis_index.html

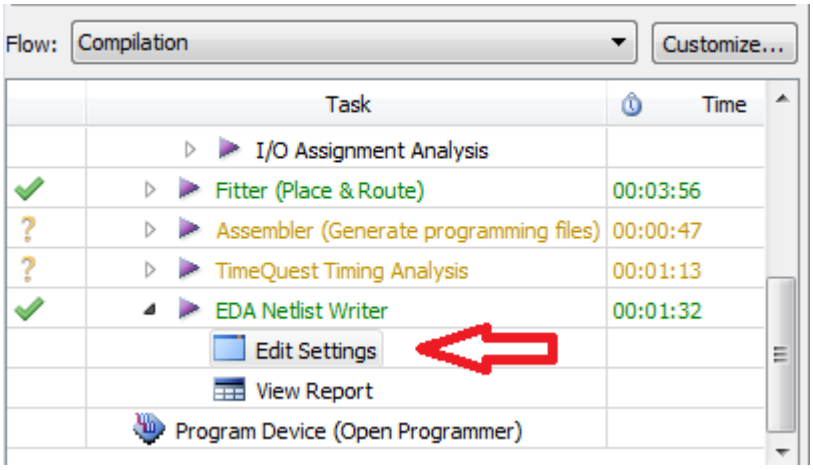
You will need to place the unzipped data from the file somewhere you can find easily from HyperLynx.

Creating the Altera IBIS File from a Quartus Project

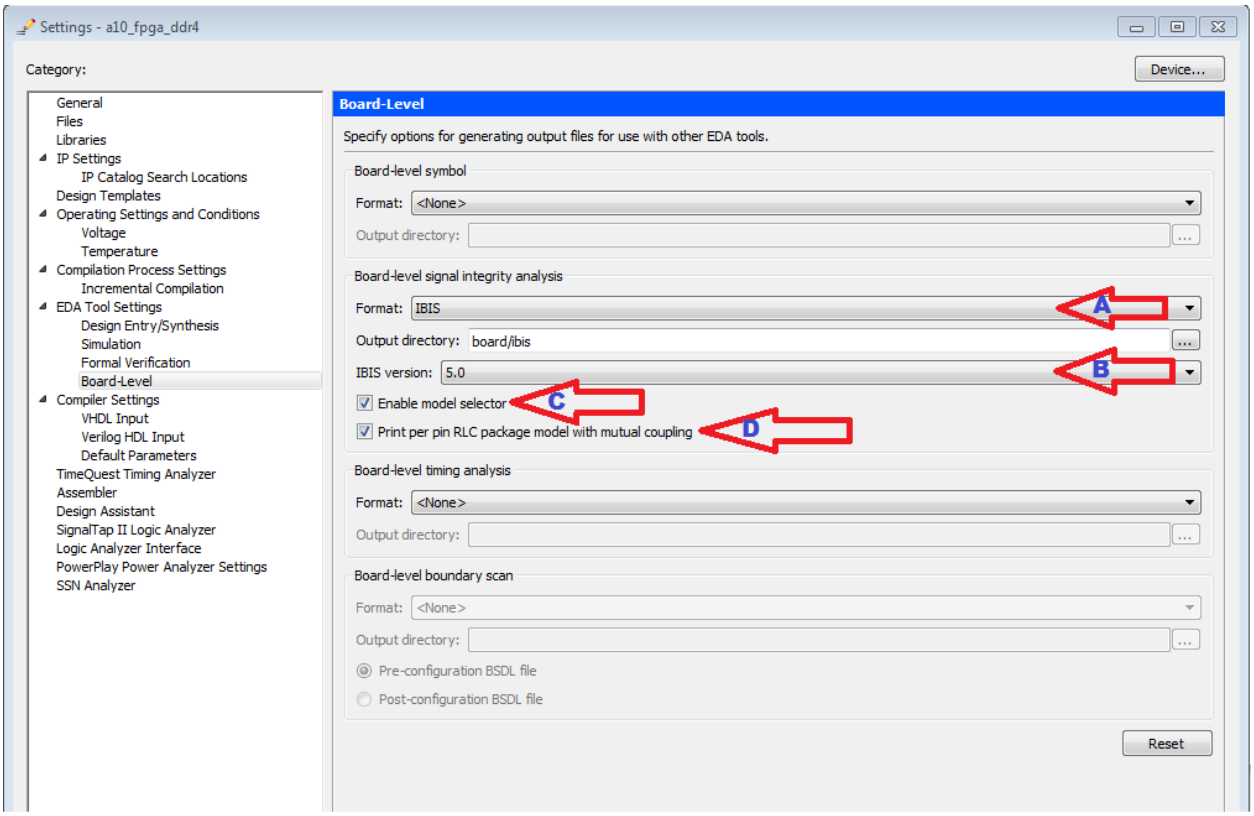
To generate an IBIS file the project is to the point where:

- A system topology has been decided
- The pin-connection guidelines applied for the project
- A Quartus® II project with the desired I/O defined
    - Often referred to as a “Golden Top” design
        - With all of the I/O assigned to various pins
        - And the pin characteristics assigned and saved in the .qsf file for the project
        - Use the latest version of Quartus® II to insure that the latest IBIS models are generated

From the Quartus® II project, under the compilation flow, select the “EDA Netlist Writer” section and double click on “Edit Settings”:



Select the IBIS format, the latest IBIS version, enable the model selector, and the per pin RLC model



Where the selections for each arrow in this diagram are:

A. Format of the output file, one can select HSPICE or IBIS, for our us it is IBIS
B. IBIS Version: using 5.0 allows per pin RLC package model generation with mutual coupling
C. Enabling model selector creates an IBIS files with multiple models for what-if simulation
D. Per pin RLC package models deliver accurate package data

The resulting IBIS file, found in the directory for the Quartus project, will contain all of the models needed for successful pre and post layout simulation.

Memory Vendor IBIS Files

IBIS simulation files are available from some memory vendors. These are usually generated from SPICE models and correlated with the physical device characteristics. The characteristics for devices from different vendors will be different.

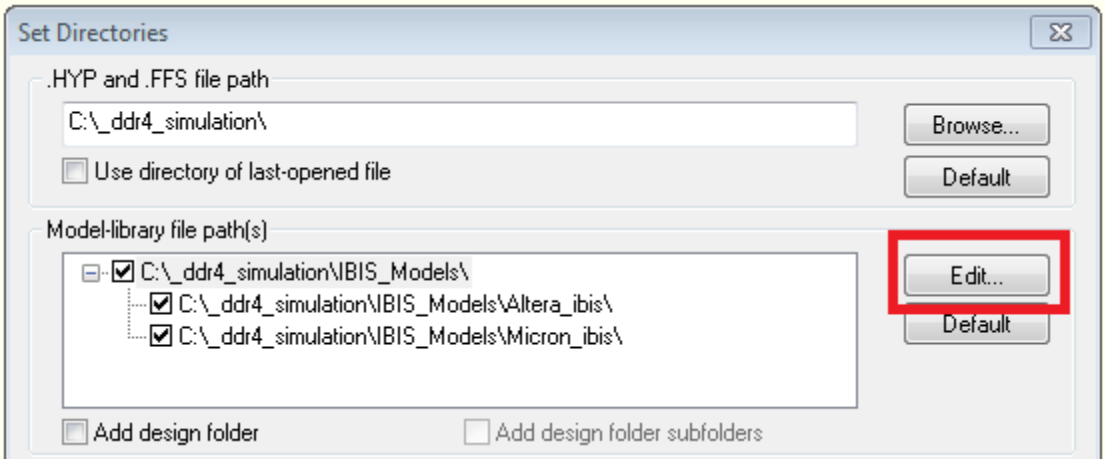
EBD Simulation Files

EBD files are available from some memory vendors. These are wrapper files that are used for boards in a system, such as DIMM modules.

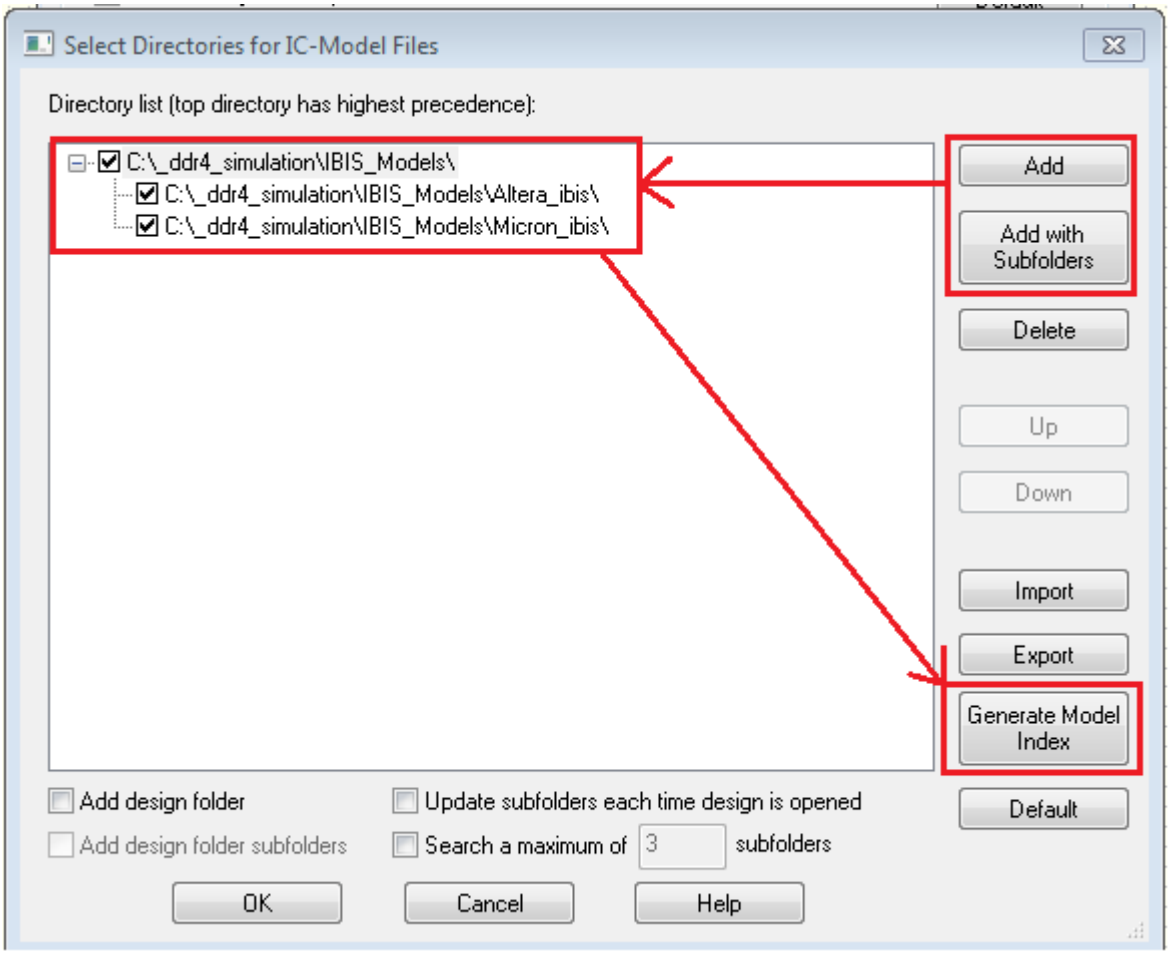
Chapter 3: Setting up HyperLynx

Locating Model Directories

Once HyperLynx had been installed it is necessary to set up the project to point to the models being used. The project can be created anywhere accessible by the computer being used. The IBIS and EBD models for the devices to be used for the project can be anywhere also. What connects them together is the assignments made in HyperLynx using Models -> Edit Model Library Paths... which gives the opportunity to indicate where the model files for the project are located. It is a good idea to start with as many models as you can get from whatever manufacturers you have for the project to avoid having to go out and search for them in the middle of assembling a simulation topology.



Click the Edit button to assign model directories to the Model-library



Add folders and sub-folders where the models are located. The Add buttons open an explorer window to let you choose the ones you want. Clicking the Generate Model Index button loads the list of models into the project and returns to Set Directories.

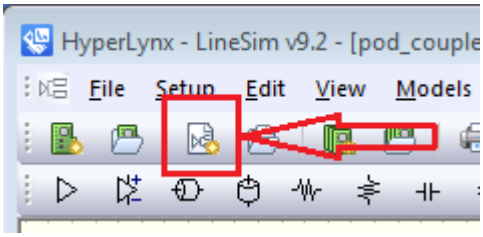
Stackup Entry

Next, set up the stackup with the materials you will be using. The graphic, above is about the minimum.

The remainder of the pre-layout section of this document will use the above stackup.

Opening a new schematic

Now a schematic can be created. Clicking an icon on the task bar opens a new, free-form, schematic.



You can place IBIS drivers and receivers, transmission lines, terminators and other topology artifacts. Examples will be shown for each topology as we go.

Chapter 4: Command/Address Simulation and Analysis

The Command/Address (C/A) system for DDR4 memories are SSTL signals. DDR4 differs from previous DDRx families in one respect, the 2T timing option for the C/A signals. This means that the data rate can be ½ of the SDR timing, allowing for a fairly long settling time when using this option.

C/A Required simulation sets

- Inter Symbol Interference (ISI) for Drive and Termination strengths
- Crosstalk for layout parallelism rules

Basic C/A ISI, channel drive and termination simulation

Setting up a sample C/A topology using the freeform schematic is easy.

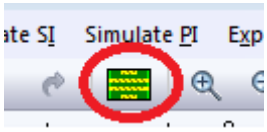
C/A Drive and Termination

For these signals the topology usually has the FPGA on one end and more than one memory device down the stream. The Fly-by topology is simple unless there are chips on both sides of the board in a "Ping Pong" configuration because of concentrated additional capacitance.

New Schematic

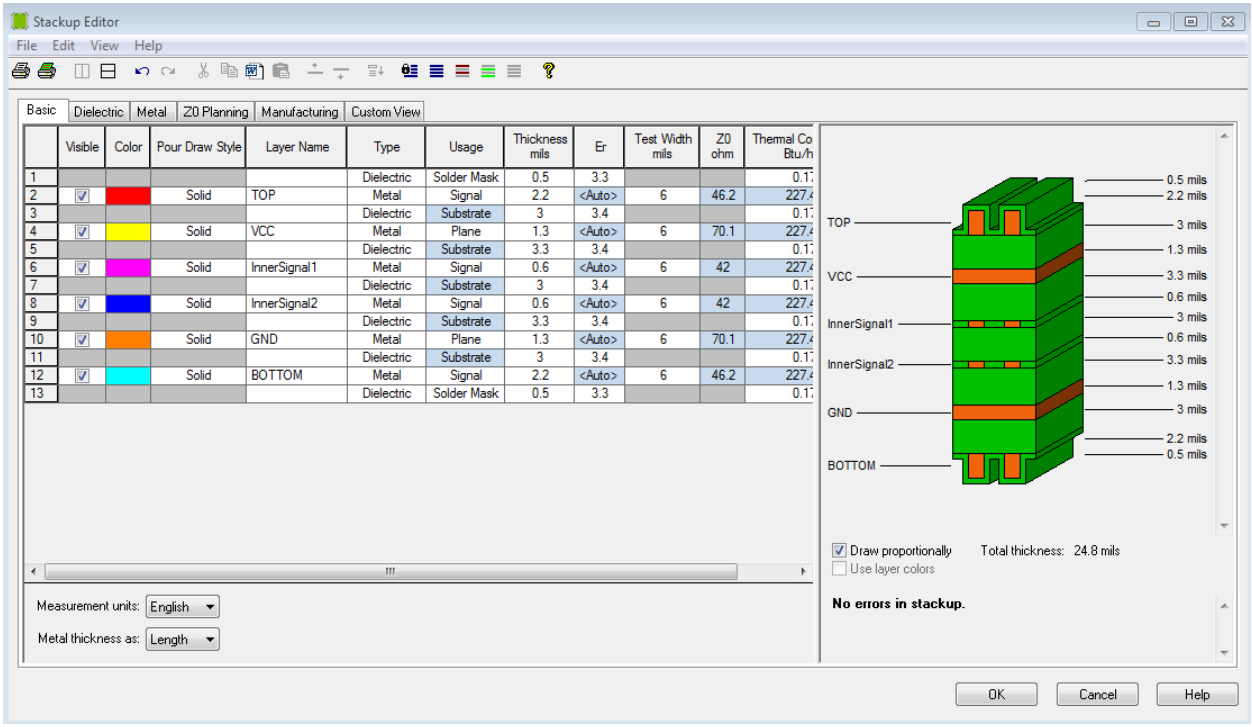
1. Open a schematic window as shown above 
2. Select File -> Save As and place the schematic in the simulation directory

Stackup entry

3. In the schematic, open the stackup 

and fill it out with the board's materials and dimensions. Be sure to use the dielectric tab to update the Loss Tangent. Then

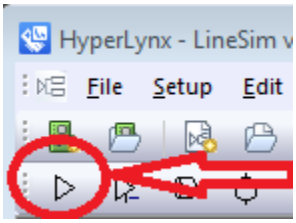
click OK to save it



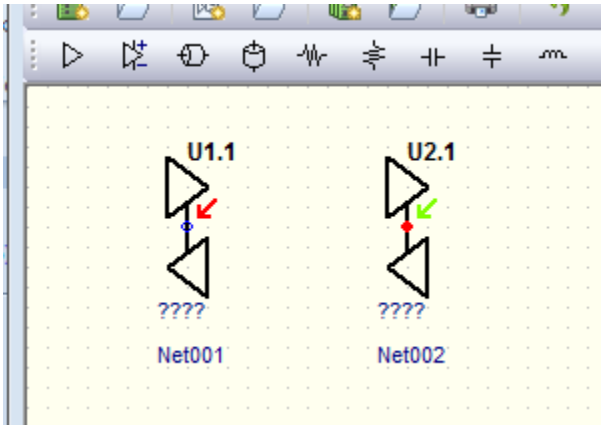
4. Since there will be more simulations coming up, use Setup -> Stackup -> Export to save the stackup so you can import it into a new schematic and save having to re-enter stackup data

Creating the topology

1. Add components to the topology by clicking on the driver symbol on the ribbon

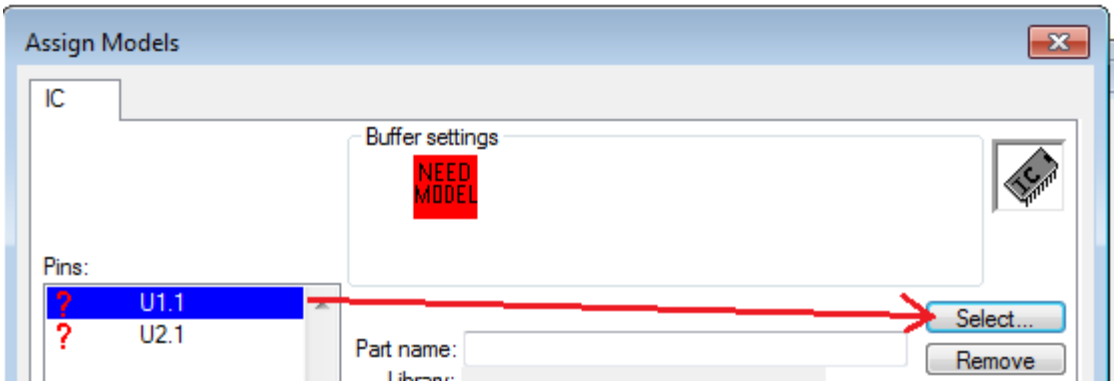


Add a symbol for each type of component in the topology being simulated

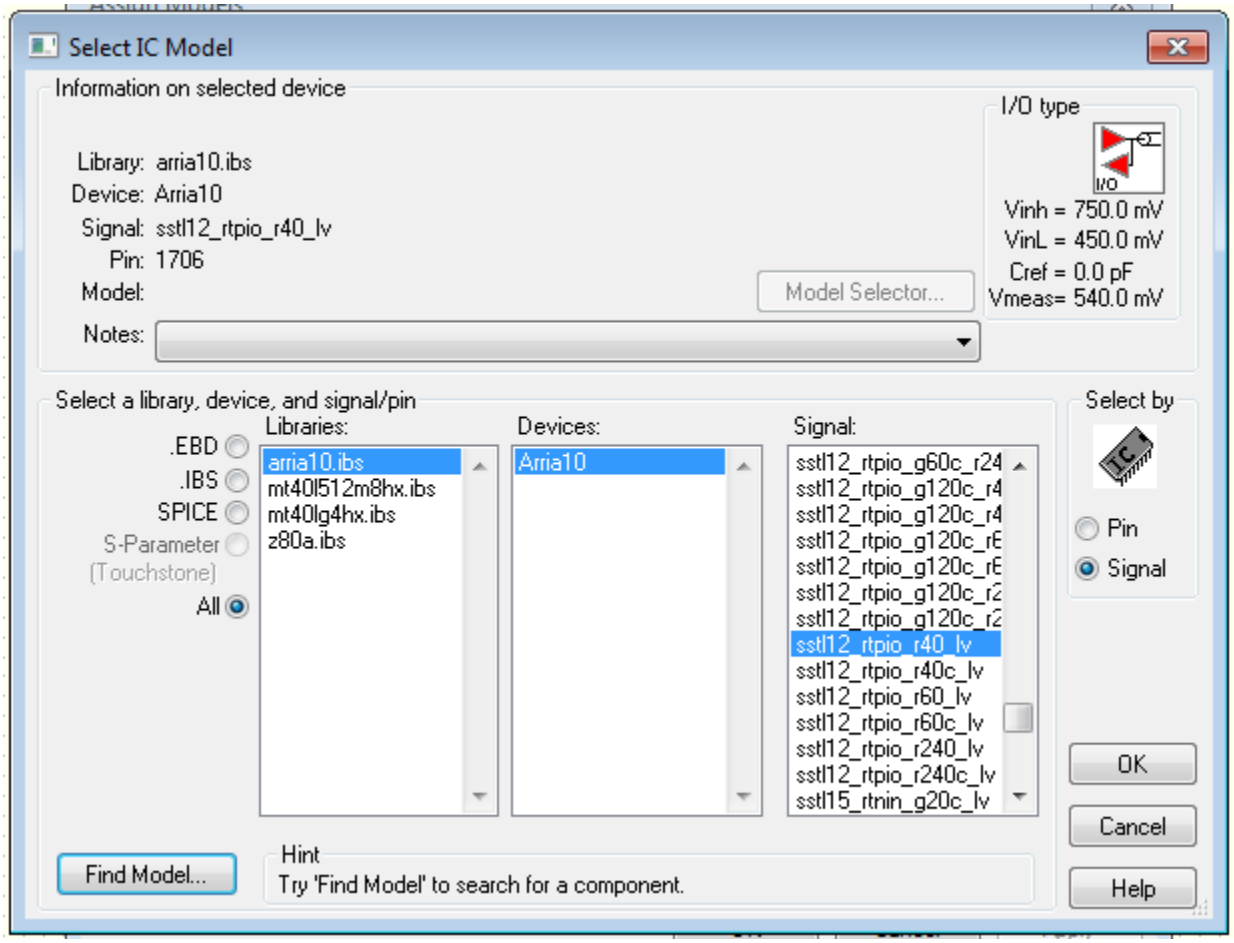


2. Double click on U1 and select the driver for the topology. NOTE: It is a good idea to open up the IBIS file, arria10.ibs, and read the notes section be familiar with the notation used for driver

capabilities

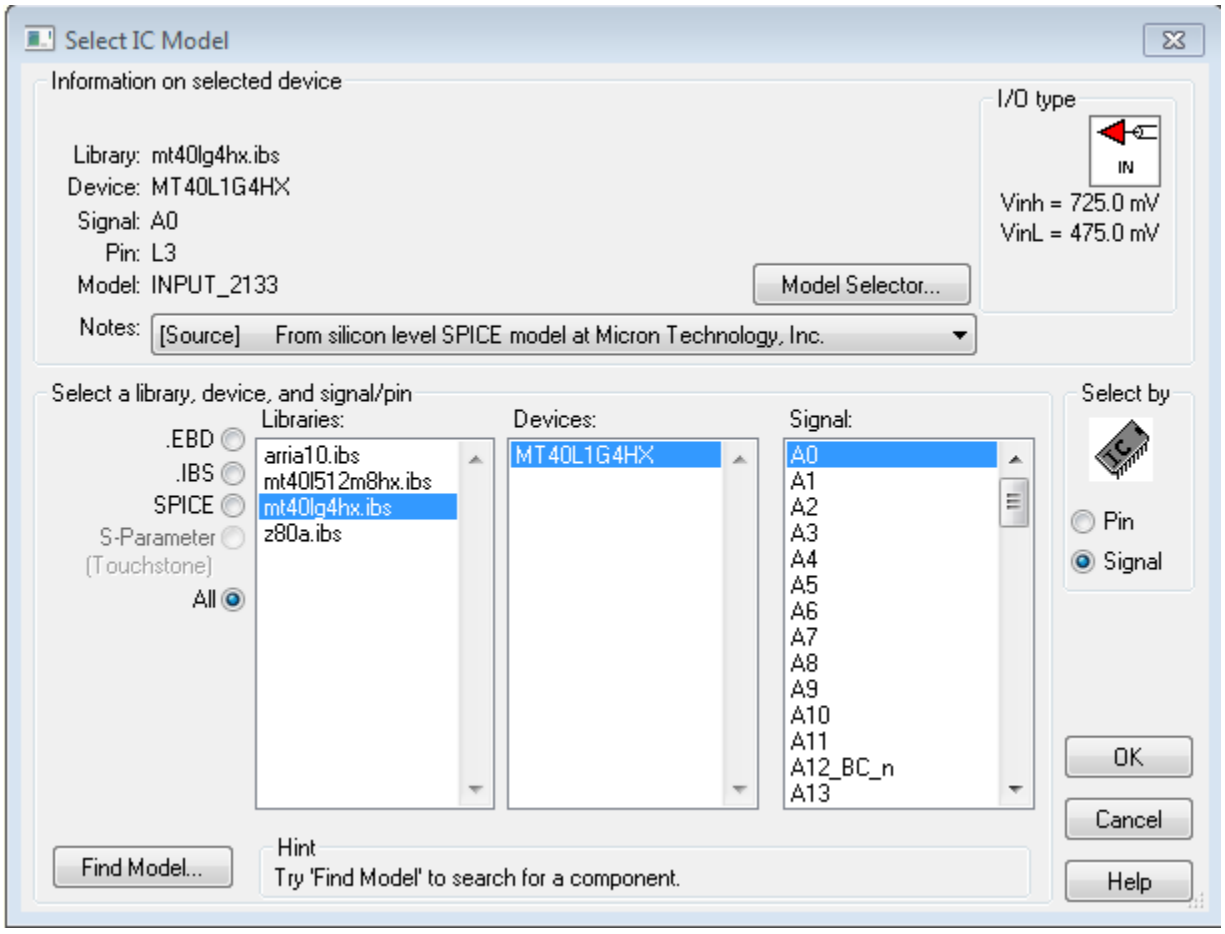


3. Select the desired driver

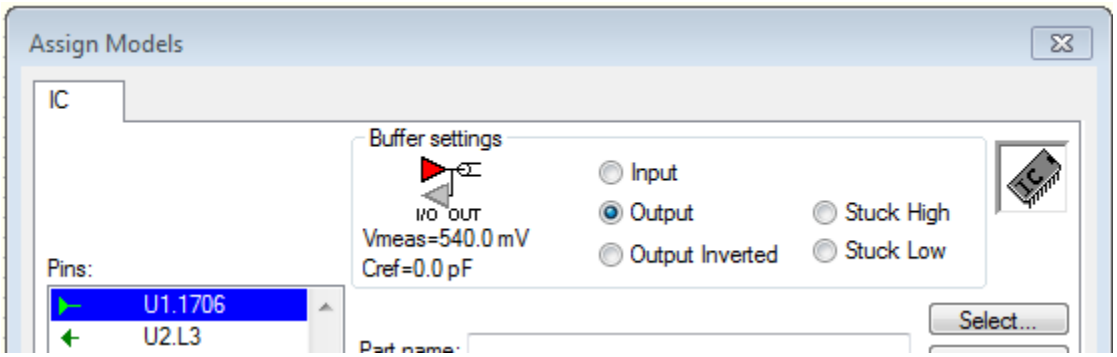


4. The typical RLC values for all pins in this IBIS file are: 289milliohms, 2.16nH, and 1.43pf.

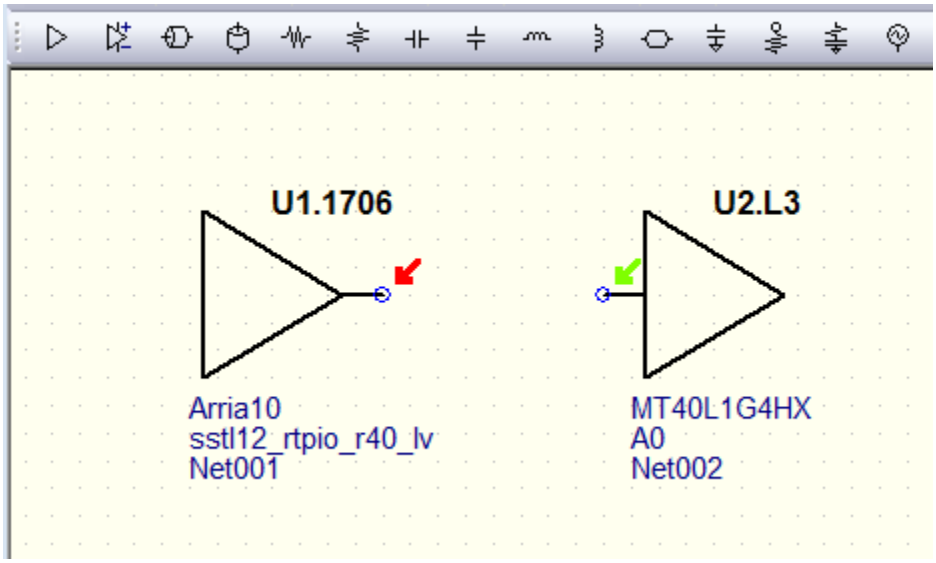
5. Repeat for U2 with the desired receiver



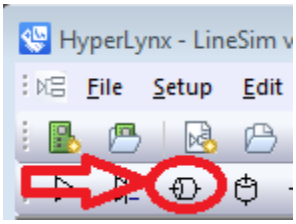
6. The RLC values for this pin are: 191.0milliohms 1.111nH 0.299pF, these values will be slightly different from signal to signal
7. Set U1 up as an output



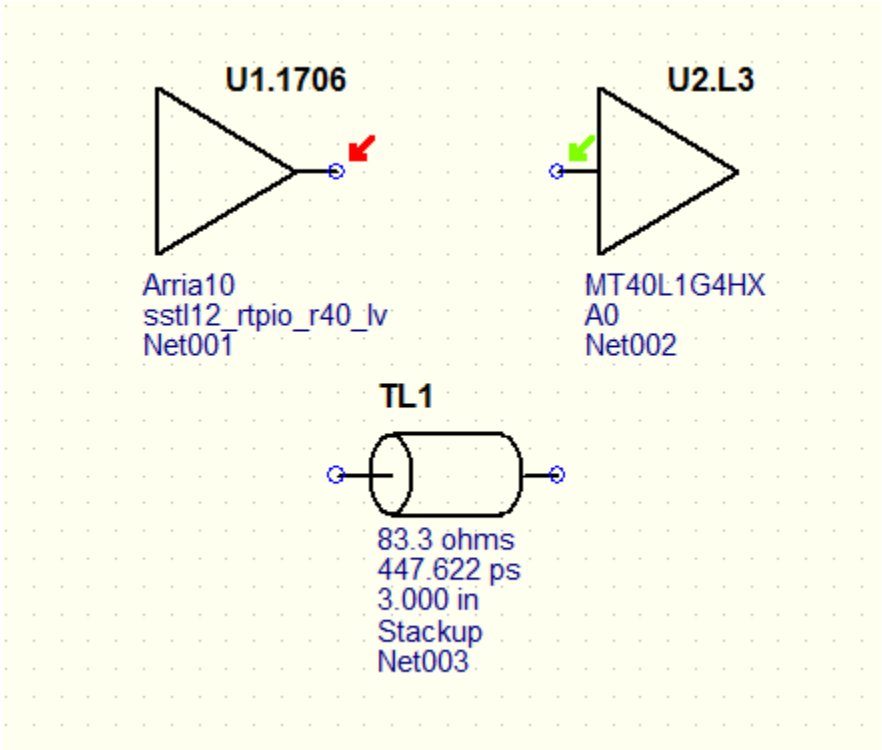
8. Click OK to return to the schematic where more accurate symbols have replaced the originals



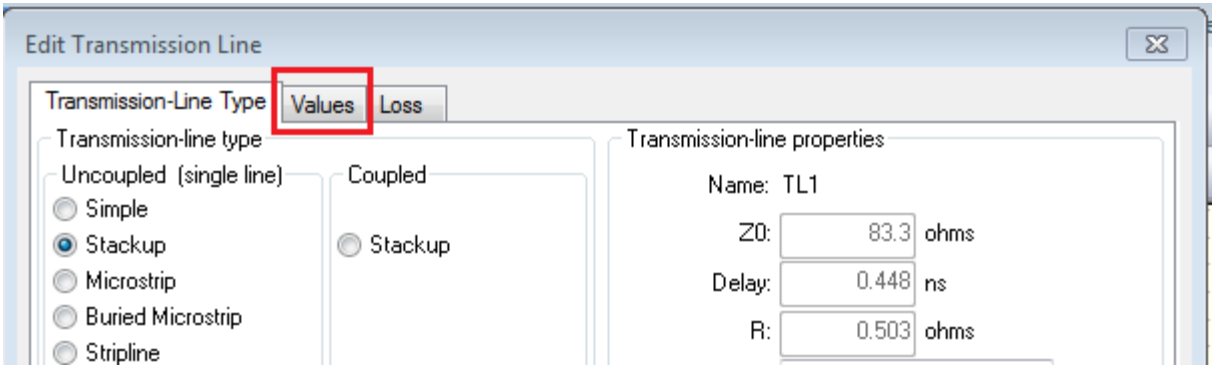
9. Next, add transmission lines to the topology for the interconnect by selecting the transmission line symbol



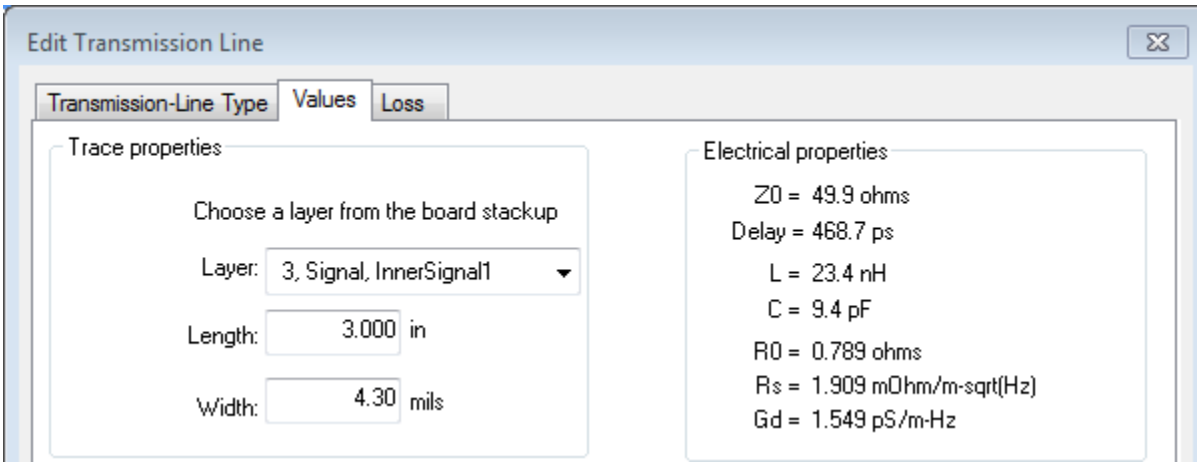
10. Place one on the schematic



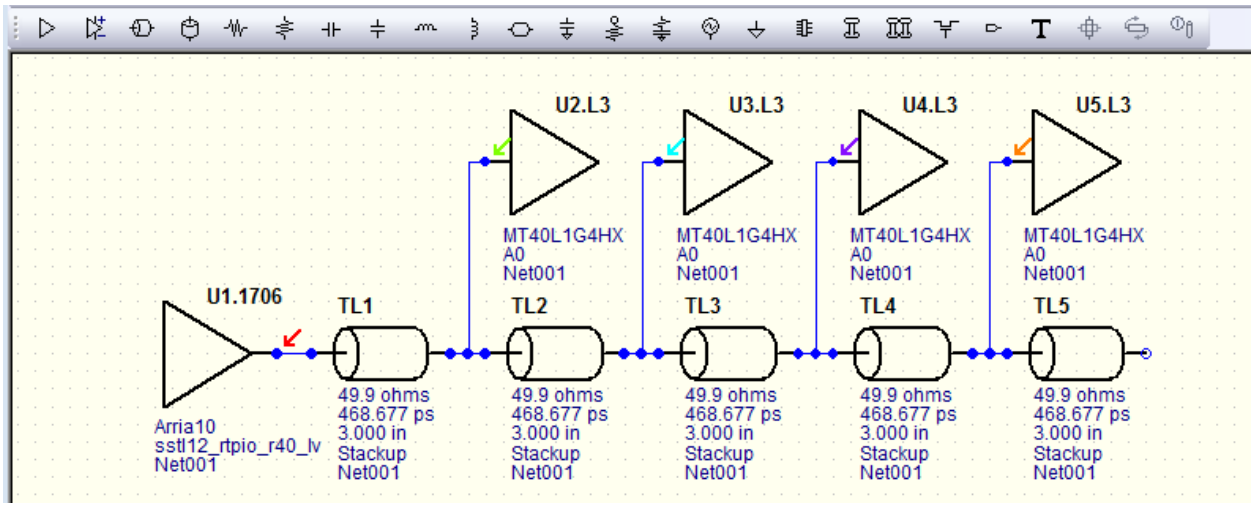
11. Double click on the transmission line to edit its properties



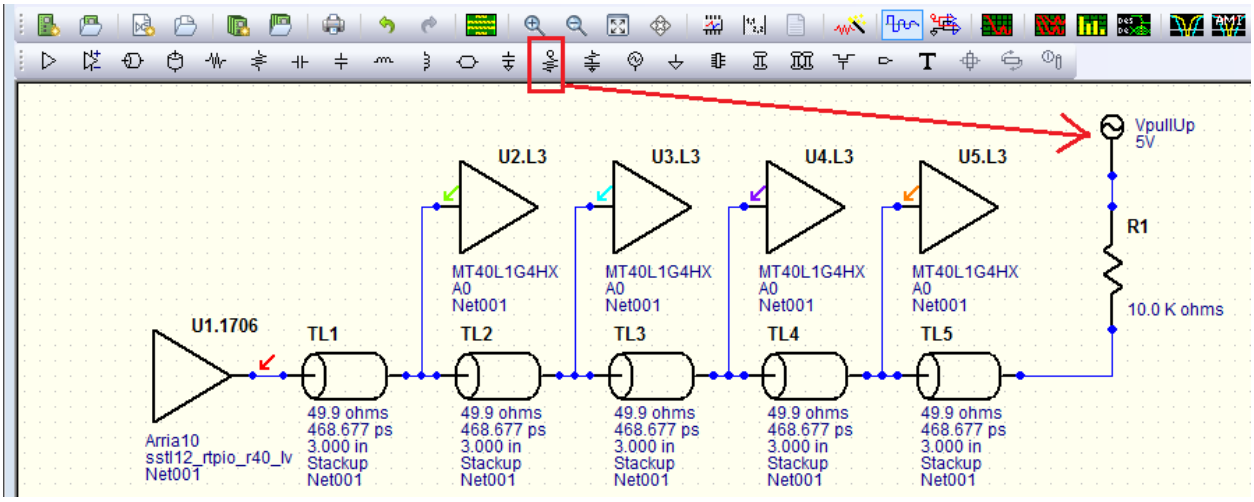
12. Click on the Values tab then place the trace on the proper layer and adjust the linewidth get the system near your target impedance



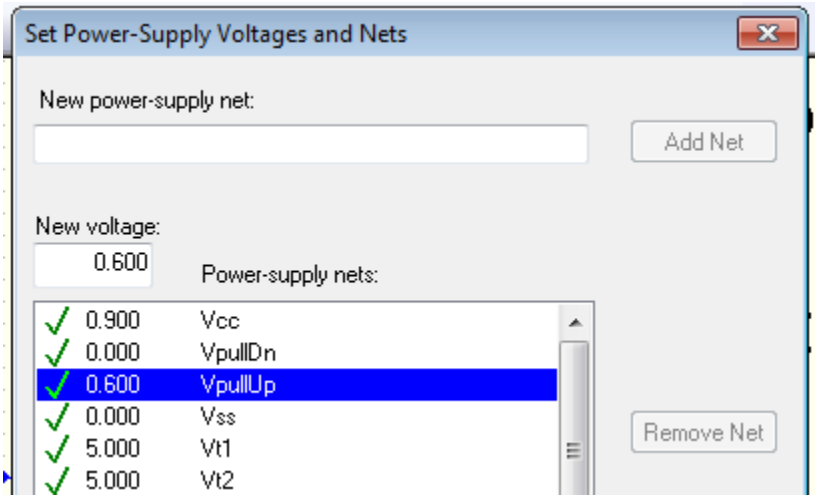
13. On clicking OK the dialog closes and the symbols can be copied and arranged to fill up the topology, in this case an array of four memory chips. Connection are made by clicking on a symbol pin and dragging to another pin



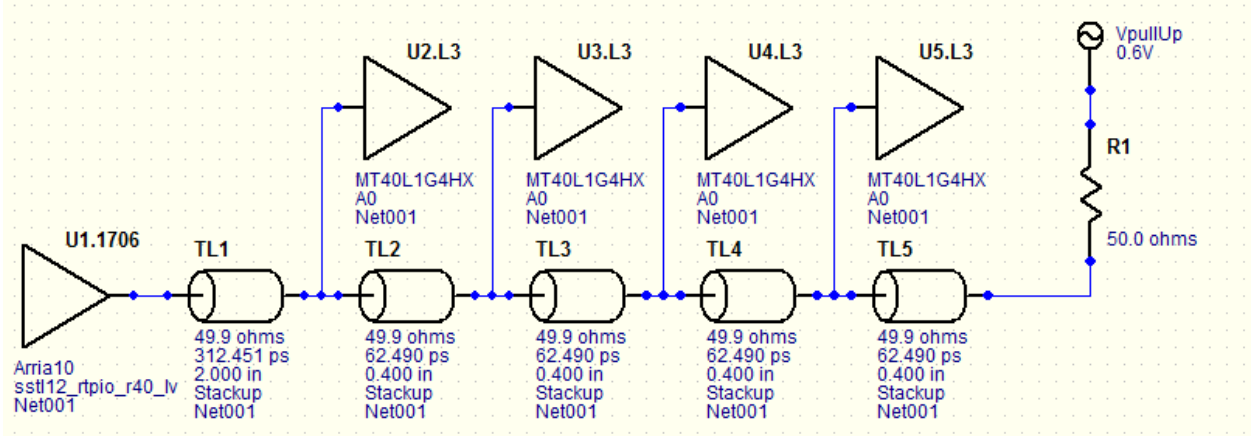
14. A termination needs to be added to the topology and wired in



15. Double click on the termination resistor and assign 50 for the value, then go to Setup -> Power Supplies and change VpullUp to 0.6 to get the termination values correct

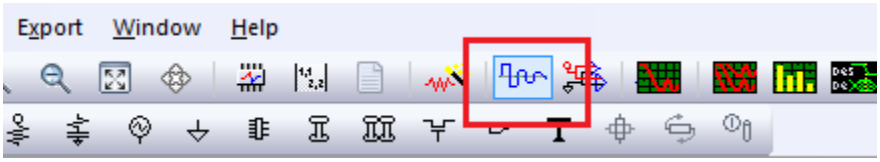


16. Double click on each transmission line segment and set the length value in the values tab to those expected for the topology



Simulation lossy channel setup

DDR4 memory systems operate at high speed and the drivers have fast slew rates. It is important that pre-layout simulations be ran with realistic material callout and lossy simulation enabled. Be sure that the Enable Lossy Simulation icon is blue before continuing.

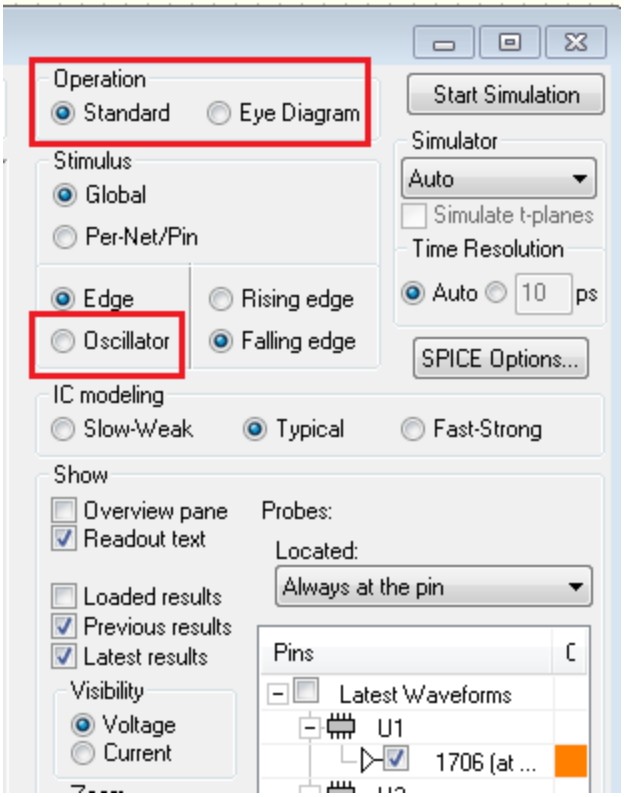


The simulations will be for a memory clock rate of 1066MHz.

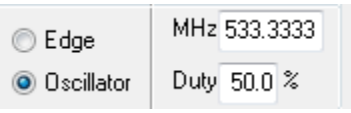
Nominal single channel ISI simulation run

This fires simulation will be done using just the single set of parameters entered so far. To start with either select Simulate SI -> Run Interactive Simulation or click on the single waveform icon  on the task bar. Here you will get an oscilloscope view with a lot of controls. The simulation that is need to see if the channel will work OK is an eye diagram. Setting this up is a little convoluted. There are a lot of choices on the right hand side of the Digital Oscilloscope. When you are setting up an Eye Diagram it is very useful to first click on the Oscillator radio button and enter the frequency for the timing. For the C/A system this would be ½ the clock frequency because the A/C system is single data rate.

Note that, under "IC modeling" Typical is selected. Altera recommends using this setting.

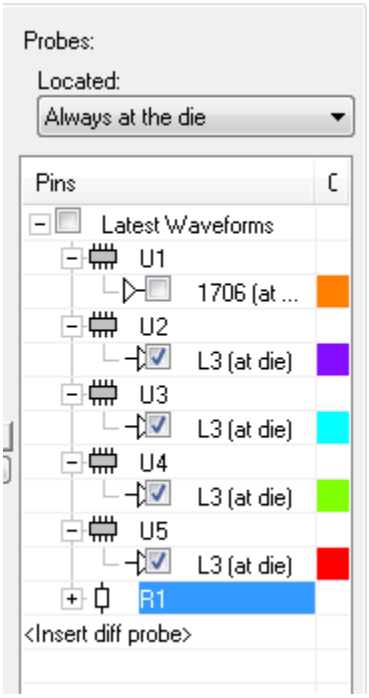


For a 1066 simulation this works out to using a 533.3333 frequency for the channel.



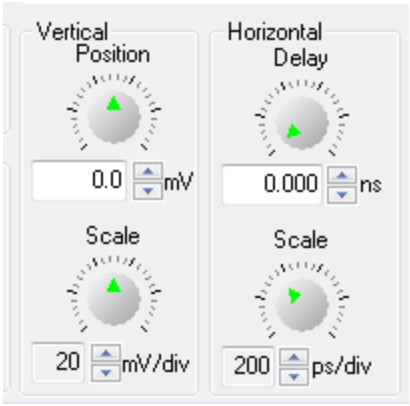
Then select the eye diagram option and the system should be set up for the simulation.

1. Select the devices you want to see the waveform for from the list. All of the inputs are of interest just now so select U2 through U5 of the latest waveforms. We also want to see the waveforms at the die instead of the pin because at these rates there is often a real difference

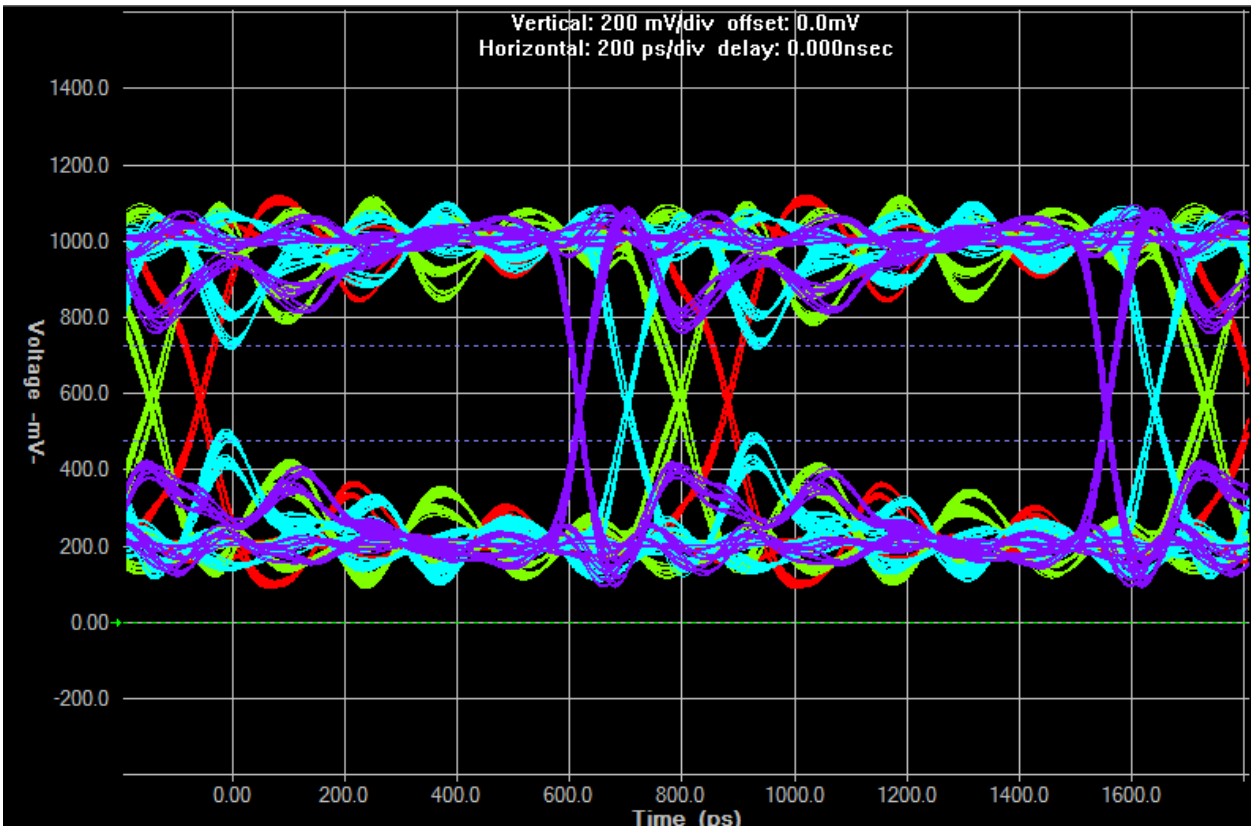


2. Run the simulation to see how things may look by clicking “Start Simulation” in the upper right hand corner of the Digital Oscilloscope. Then zoom in on the resulting eye by clicking on the zoom to extents button on the scope.

Adjust the sweep settings to something you are happy with.

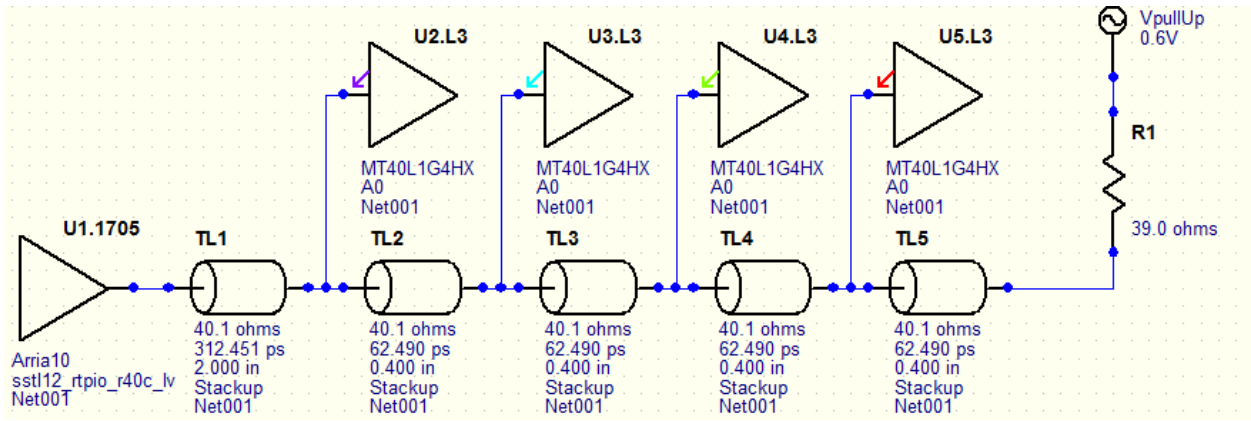


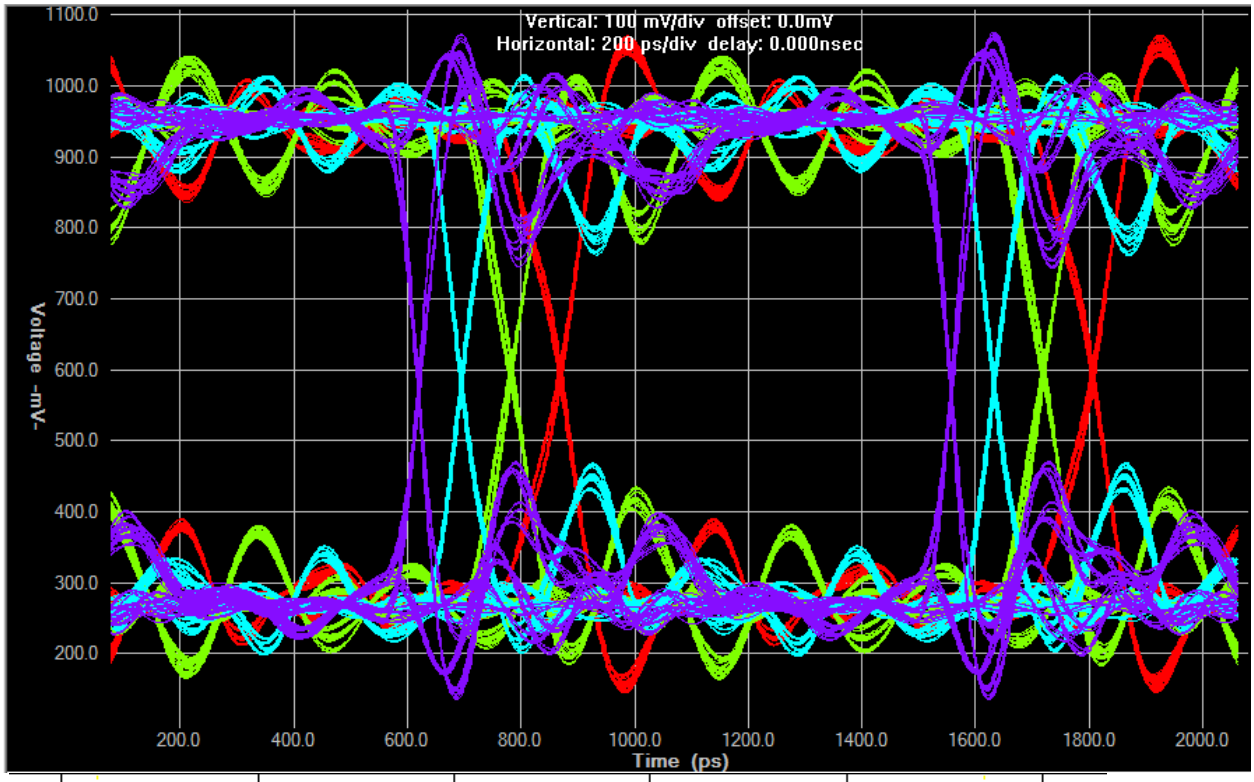
The results look something like this

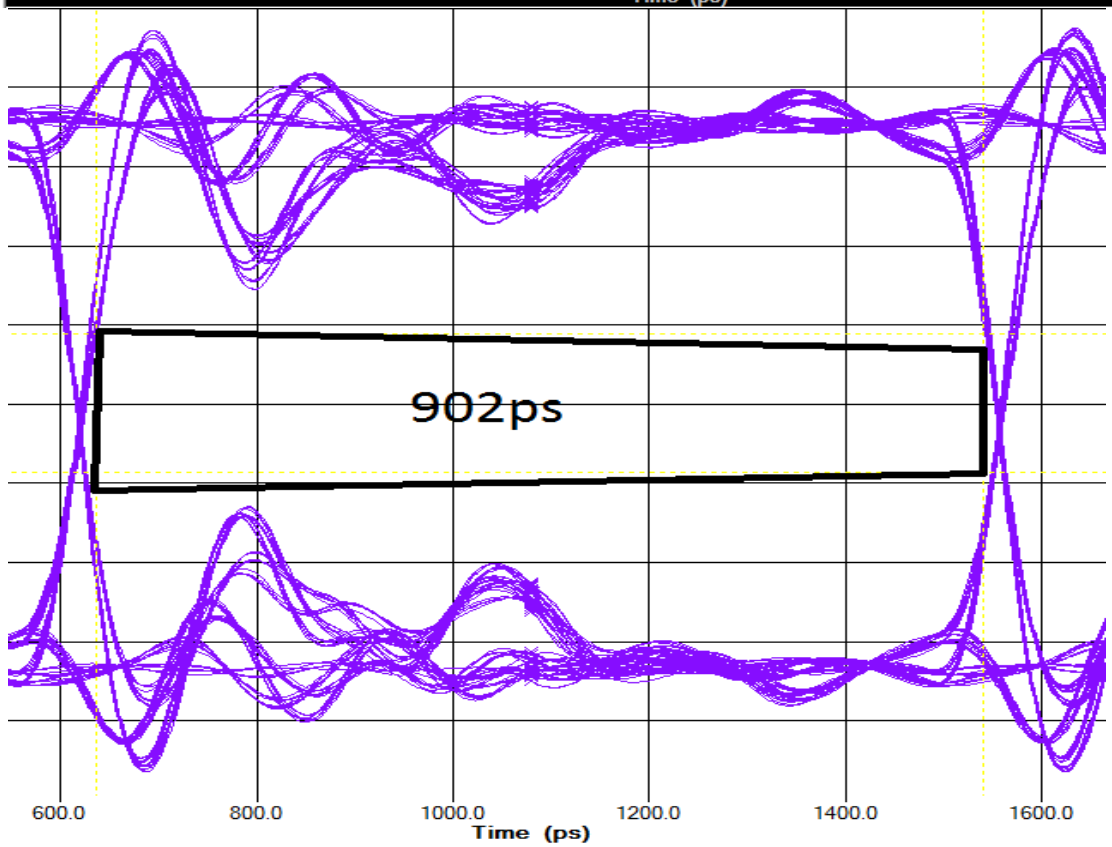


There seems to be some sort of problem here and it does not seem to be a reflection from the end of the chain. The signal at U3 actually rings back to the threshold. Though it settles later the eye opening will be compromised.

3. The only selections for the driver are 40 Ohms and 60 Ohms. We will now match the impedance of the transmission lines to 40 Ohms, the termination to 39 Ohms, and see what the waveforms look like


U2.L3
U3.L3
U4.L3
U5.L3
VpullUp
0.6V
R1
MT40L1G4HX
A0
Net001
MT40L1G4HX
A0
Net001
MT40L1G4HX
A0
Net001
MT40L1G4HX
A0
Net001
39.0 ohms
U1.1705
TL1
TL2
TL3
TL4
TL5
Arria10
sstl12_rtpio_r40c_lv
Net00T
40.1 ohms
312.451 ps
2.000 in
Stackup
Net001
40.1 ohms
62.490 ps
0.400 in
Stackup
Net001
40.1 ohms
62.490 ps
0.400 in
Stackup
Net001
40.1 ohms
62.490 ps
0.400 in
Stackup
Net001
40.1 ohms
62.490 ps
0.400 in
Stackup
Net001





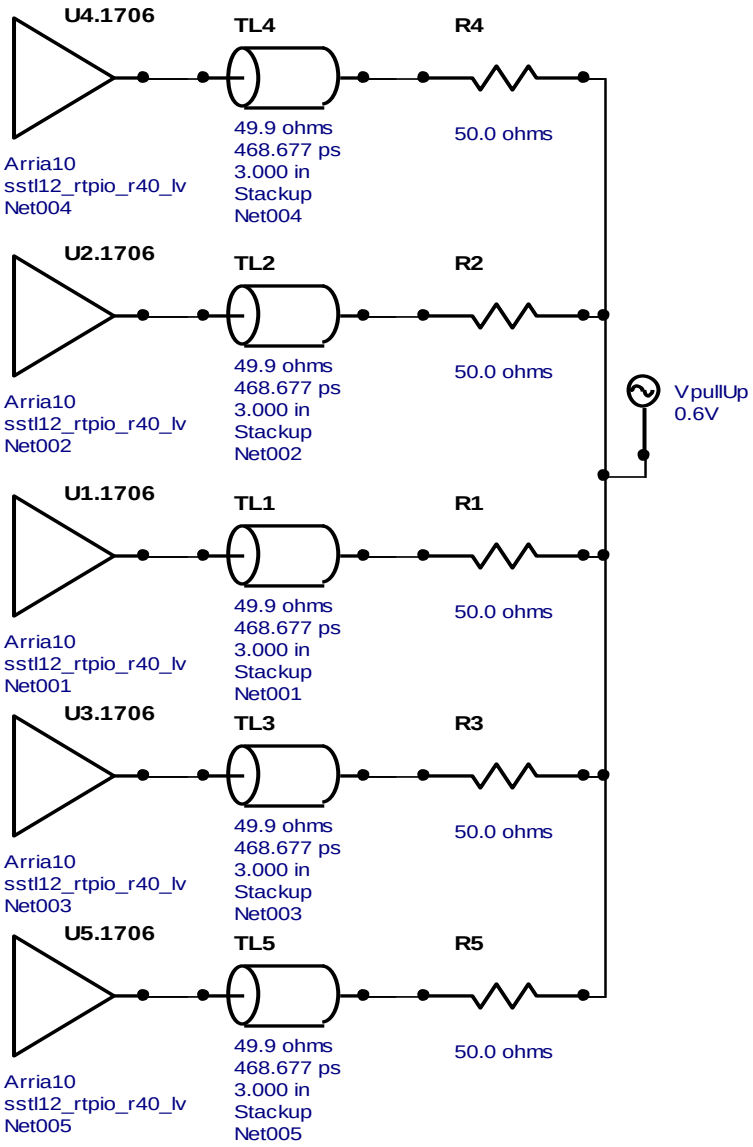
4. It looks like using A lower board impedance could be really good. There is a push to use lower impedances for transceiver channels because matching to the BGA, vias and connector cutouts

is much easier when the board impedance is lower. It is a good idea to work with the whole team when making these decisions.

Crosstalk Simulation Setup for C/A Channels for Parallelism

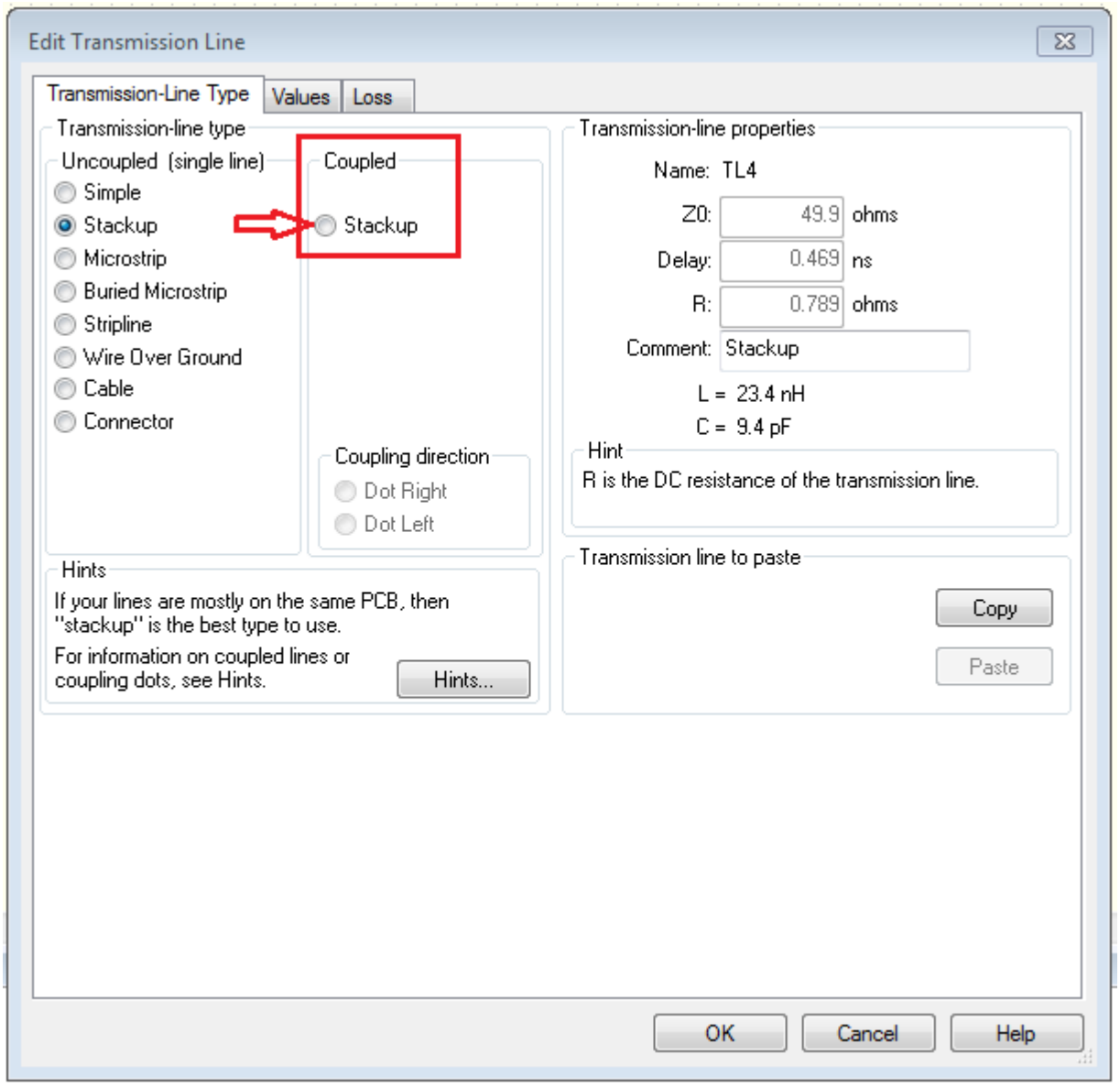
Now we will examine parallelism for the C/A channel. The victim trace will have four aggressors. This topology can be created easily from one of the above topologies. The DDR4 parts were removed and most of the T-lines.

Design File: Command_Address_prll.ffs <G:_ddr4_simulation\Cmmd_Addr\>
HyperLynx LineSim v9.2

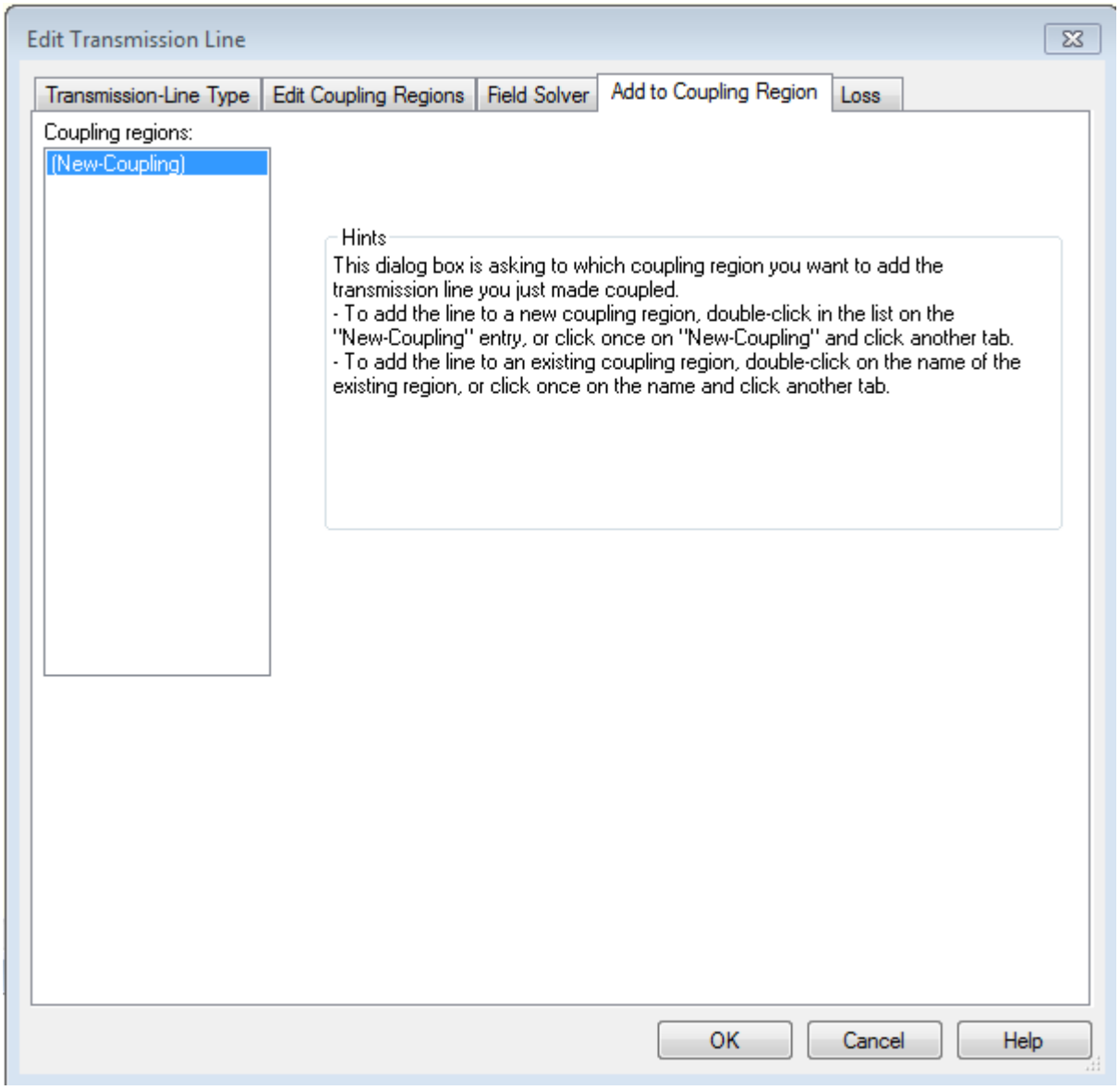


When the parts have been arrayed and connected the T-lines will be coupled together by doing the following.

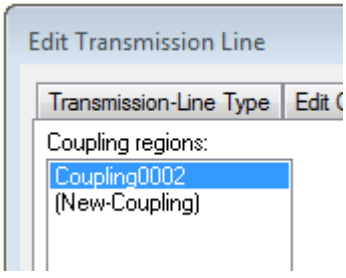
1. Double click on TL4 to open the Edit Transmission Line pop-up and then click on the Stackup option under Coupled



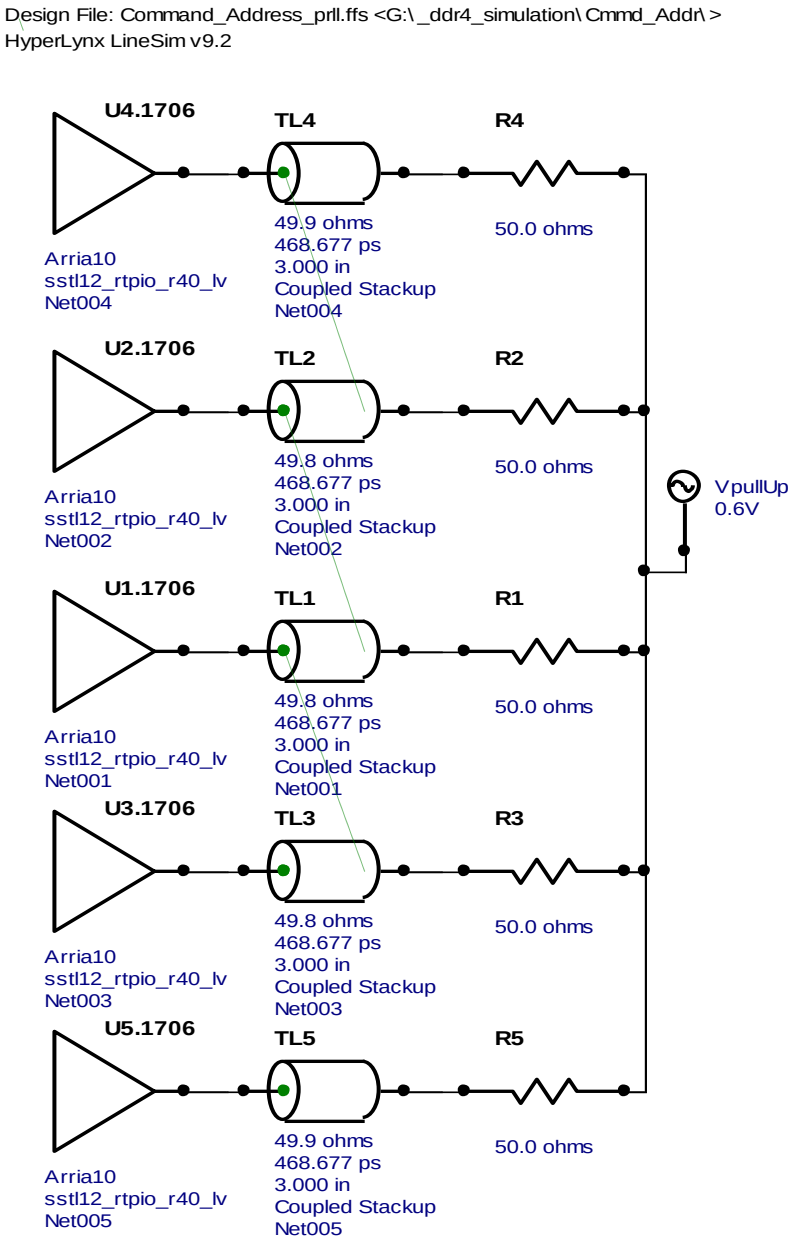
2. The pop-up will change to the coupled line version. Click OK to start the new coupled region.



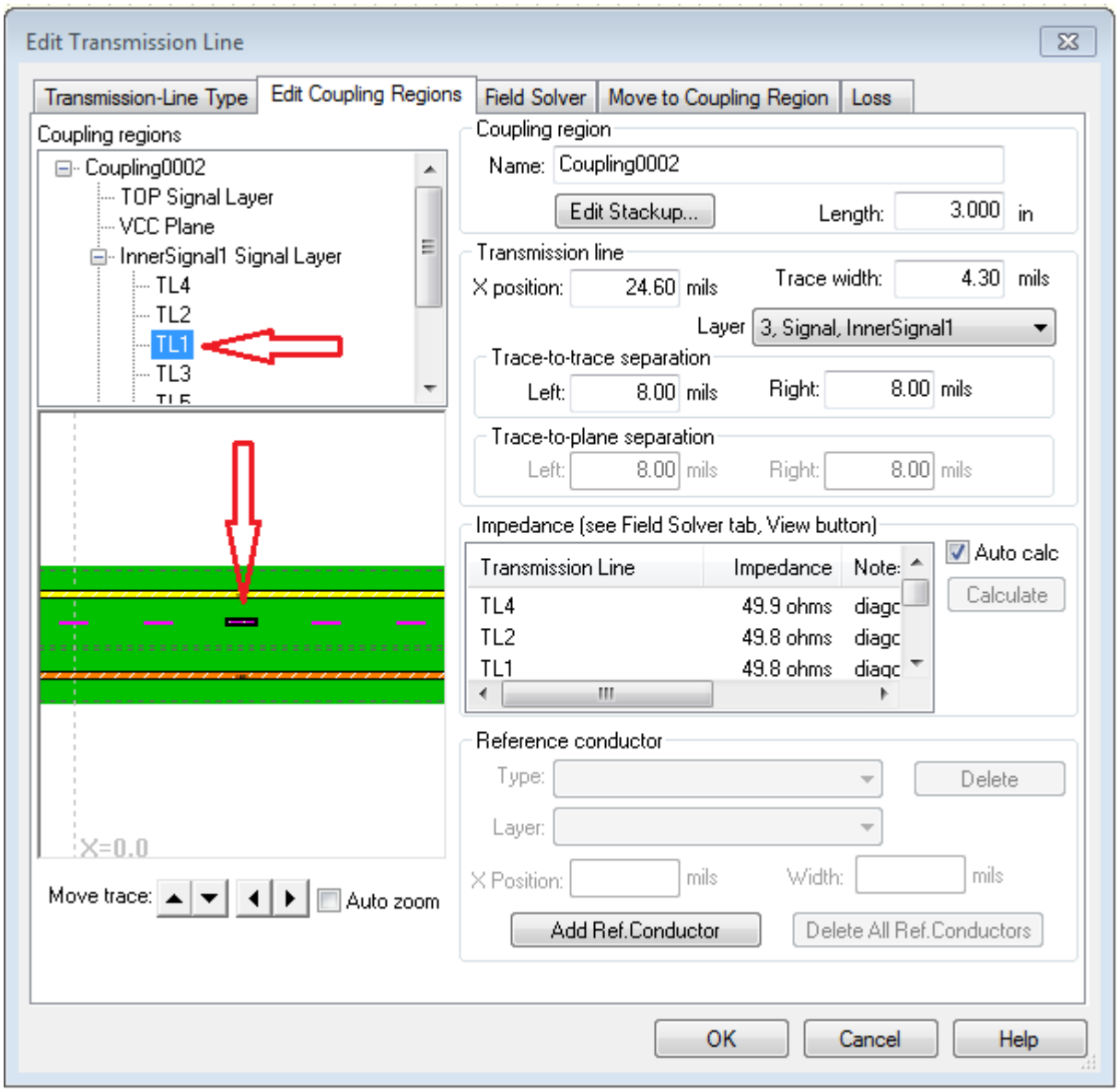
3. Next, double click on TL2 and click on the Coupled Stackup button then click OK. Do not select "(New-Coupling)" as we are adding the transmission line to the existing coupling.



4. Repeat the above sequence, progressing down the array of T-lines to complete the topology. You will observe dashed lines indicating the coupling between the T-lines.



5. Double click on any T-line to observe the position it has in the array for simulation.



There is a lot of information about the array in this tab of the pop-up. Observe TL1 is selected and the position of TL1 is shaded in the physical view.

1. Close the pop-up

Crosstalk simulation theory

Now we will analyze the coupling between the C/A signals. Before you enter into this effort, a decision needs to be made concerning the goals of the process. The C/A section performs like a source synchronous interface, everything happens at fairly close to the same time. Transitions of these signals from high to low and low to high occur very close together in time. After this simultaneous switching event it is very calm. The transitions get a little messy then the steady state of the eye damps out to a very good opening. If you are designing a board for the PCIe form factor and, therefore, need to pack

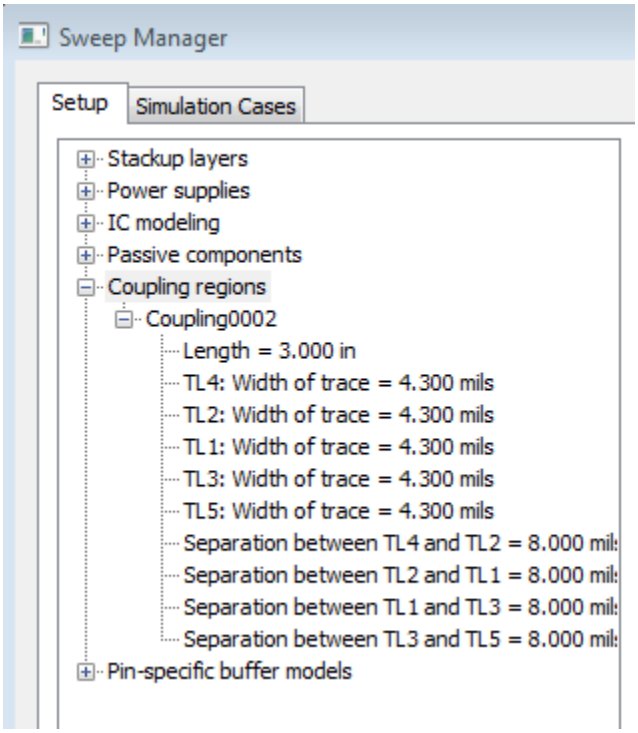
the interface into a narrow channel on a crowded layer, then your needs are not going to be met for someone designing a signal integrity demonstration board with almost infinite space. Many high performance memory systems have been designed with a -20db crosstalk level for the C/A signals. Signals outside of this grouping may deed more isolation due to the possibility of transient events after the C/A signals should be settling. For this reason, the following investigation will be focused on a -20db crosstalk limit.

C/A Parallelism Simulation and Analysis

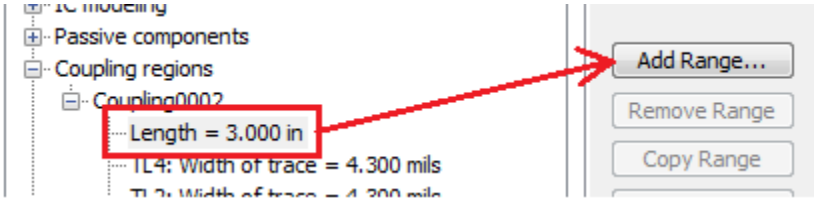
In order to analyze the coupling between the aggressors and the victim in as short of time as possible the strategy of using a sweep of parameters we can control. The parameters we can control have to do with signal trace spacing and coupling length.

Here is how to set it up:

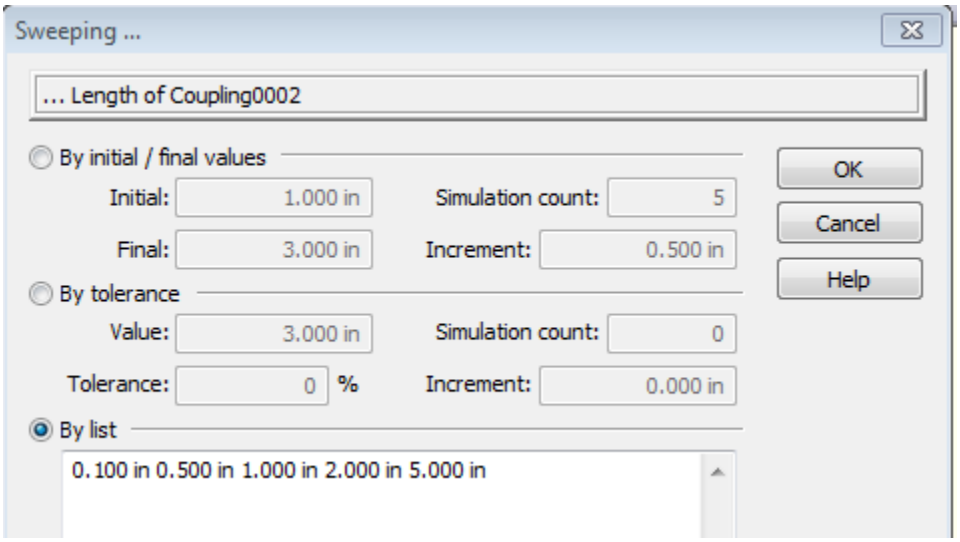
2. Select the Run Interactive Sweeps… icon  or select Simulate SI -> Run Interactive Sweeps to open the Sweep Manager.
3. Select Coupling regions and then the coupling setup where all the T-lines are



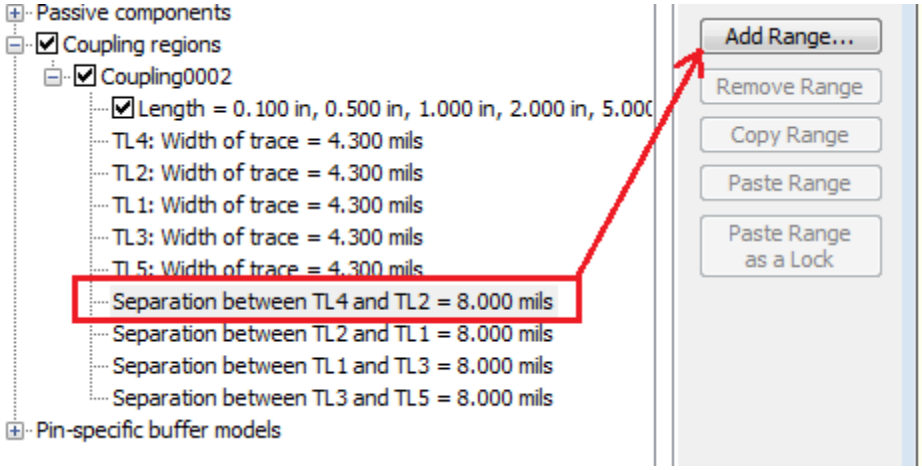
4. Highlight Length then select Add Range



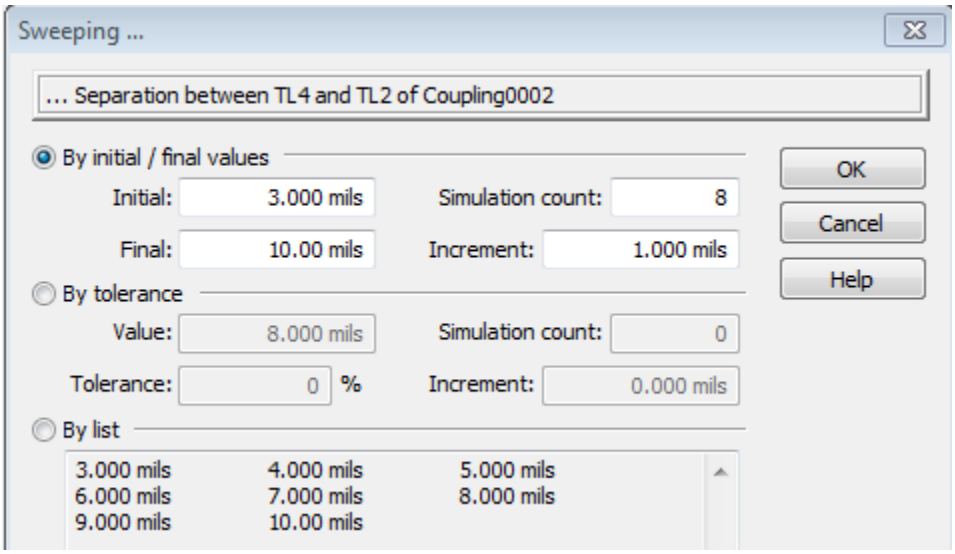
5. In the pop-up Sweeping window select By list and enter values for the breakout and some other distances up to the longest expected trace length. You can run many simulations easily but it becomes a problem of managing the data.



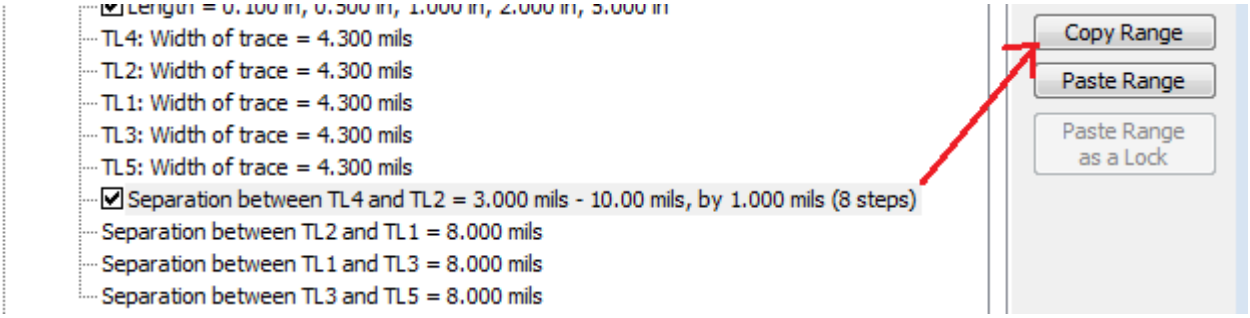
6. Next the separation between the T-lines will need to be swept. Select the first Separation between… and click Add Range



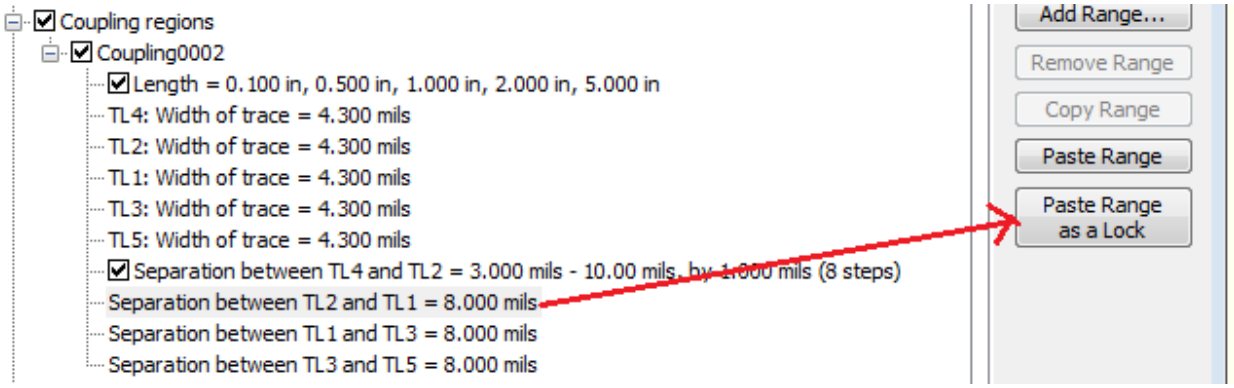
7. Enter in some spacing values and increment data, then click OK



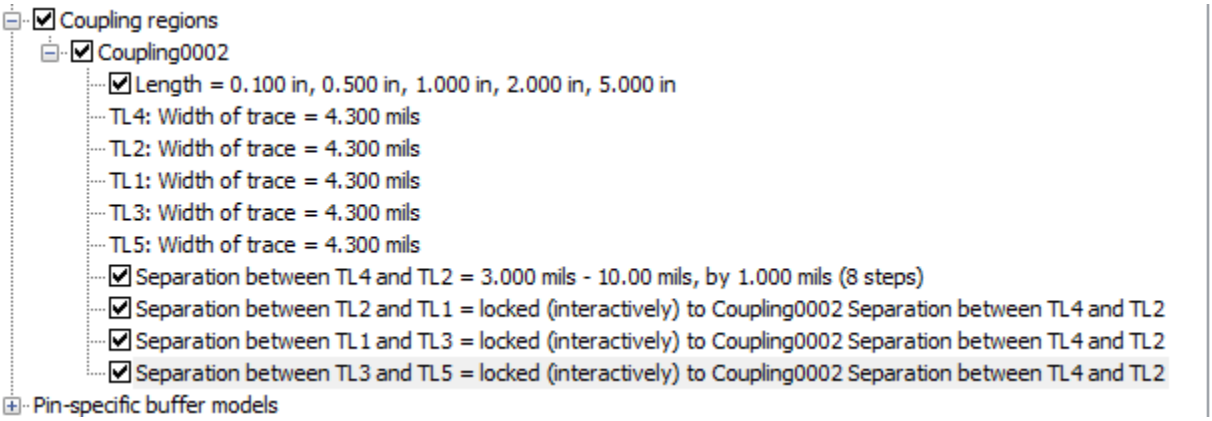
8. Now select the separation just finished then click Copy Range



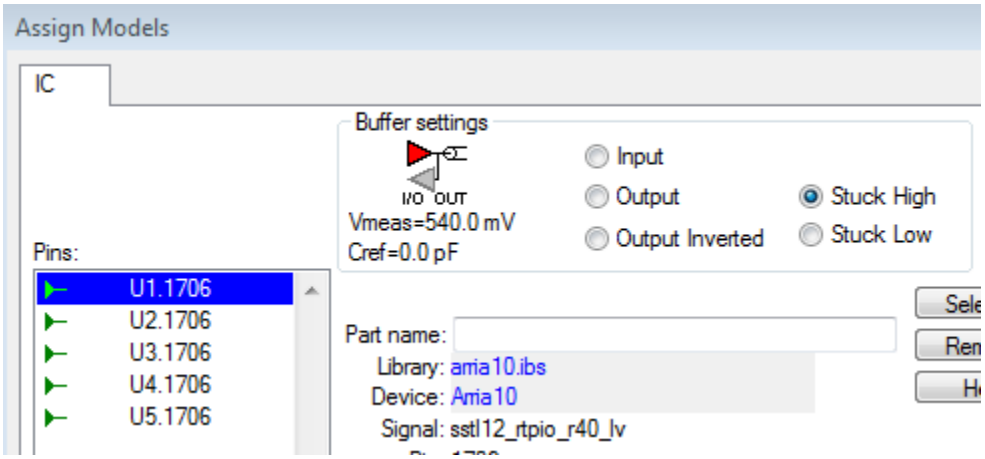
9. Then select the next Separation and click on Paste Range as a Lock to connect them together.



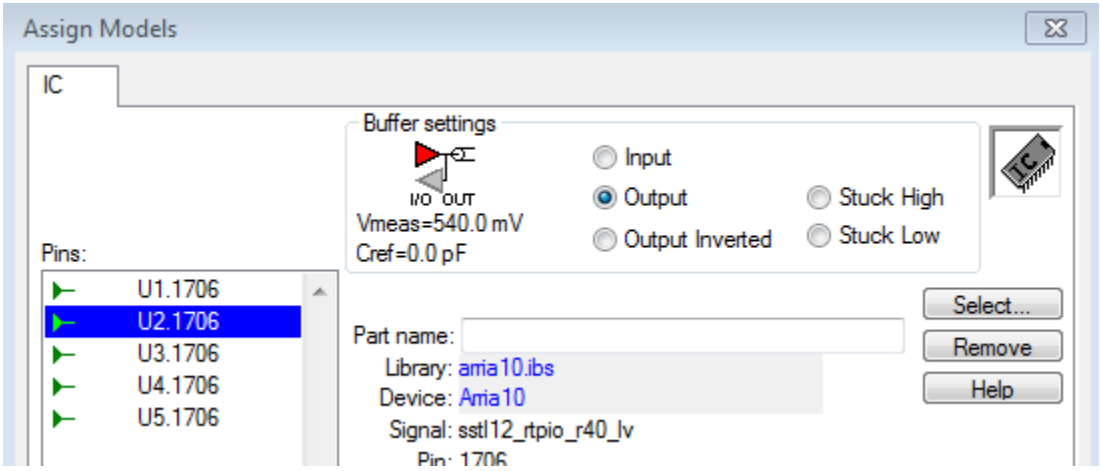
10. Repeat for the other separation selections



11. Now everything is set up except for the drivers. Close the Sweep Manager and set the driver for U1 as Stuck High

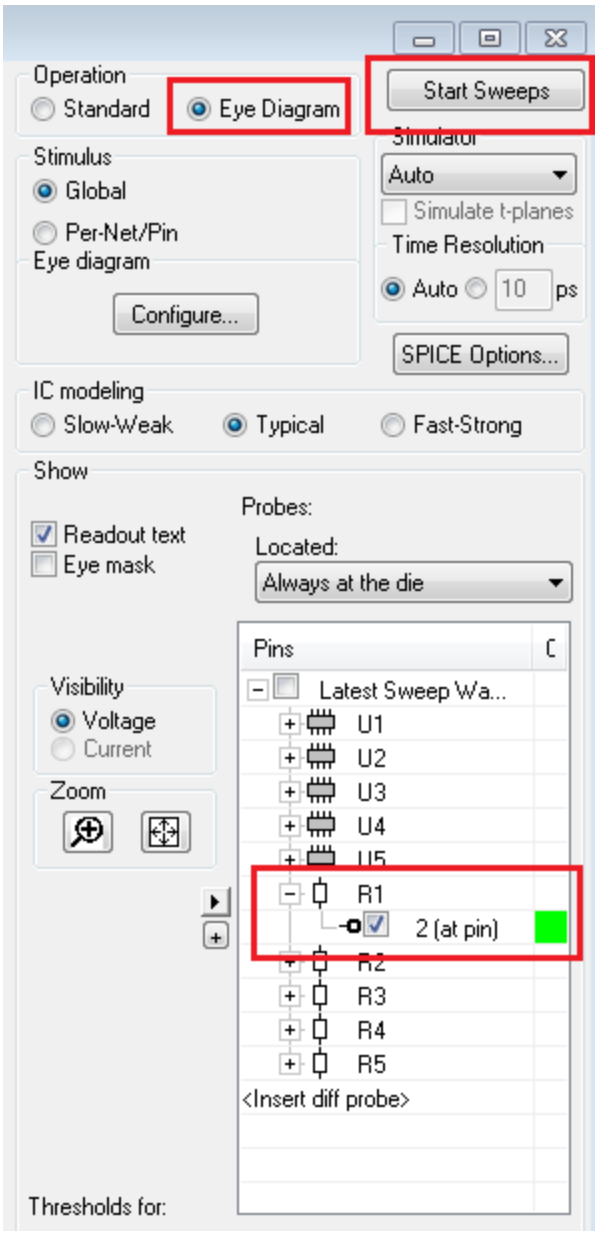


12. The other drivers are just left as Output to switch together and provide the highest aggressor interference

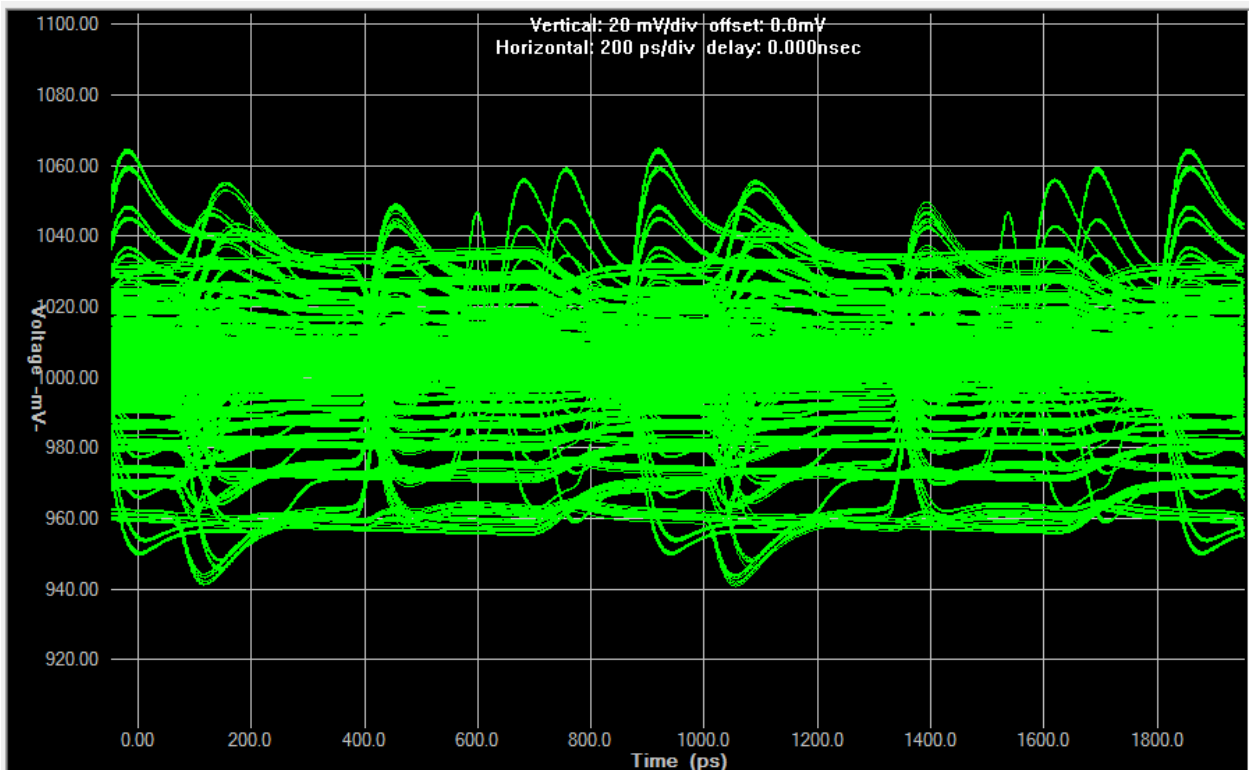


13. Reopen the Sweep Manager and click Run Sweeps. Here we will use the settings from the termination run and not worry about the frequency of the clock because the slew rate of the drivers is the determining factor for coupling. Set up the simulation for eye diagram to get some randomization of the coupling, select the termination resistor of the victim driver for analysis

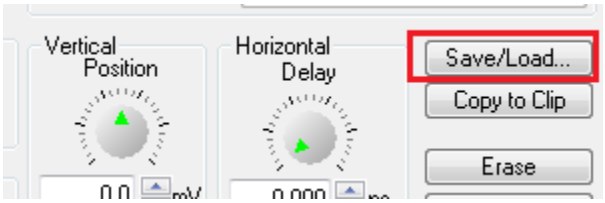
and click Start Sweeps.



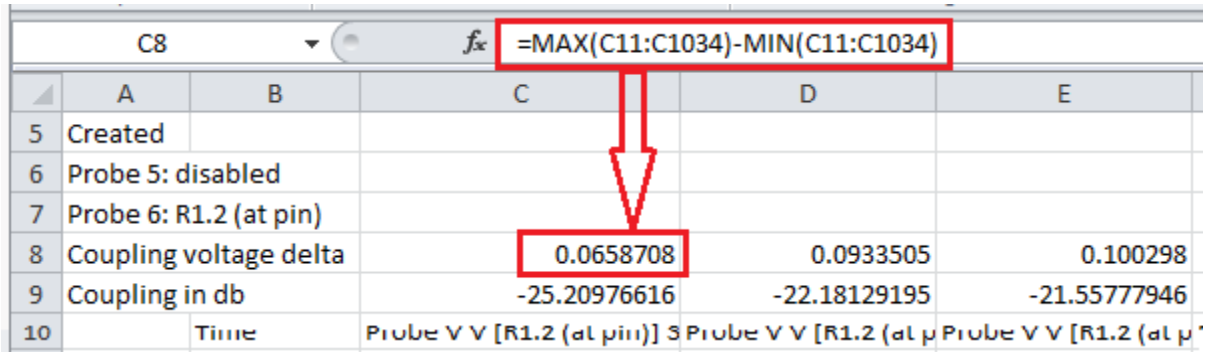
14. After the sweeps finish there should be an output that looks like this.

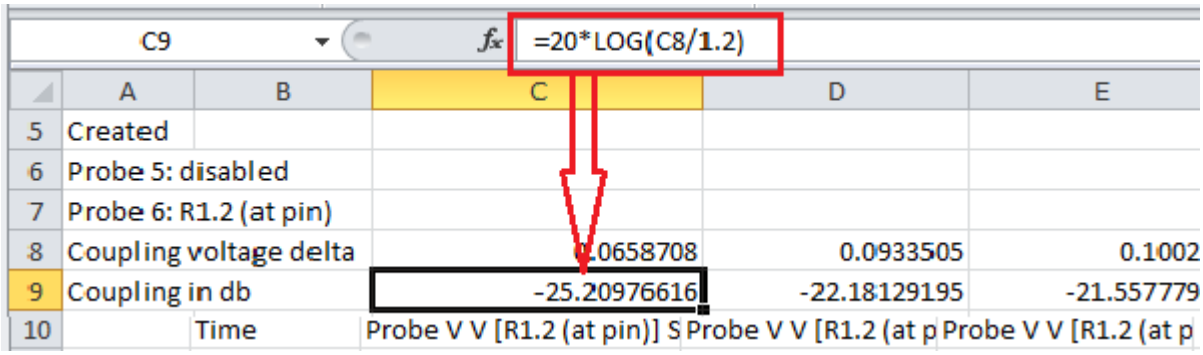


15. Now to analyze the data:
    - Use the Save/Load button to export the data to .csv file for analysis



16. Open the .csv file in a spreadsheet to enable analysis. This will be a huge spreadsheet but there is room at the top to calculate the crosstalk in db for the data. This is created by taking the maximum minus the minimum voltage for each column, dividing this by 1.2 and then calculating 20*log() of that value. It is up to the engineer to select the parallelism value for the interface. Many systems have been created using the value for -20db with good success.





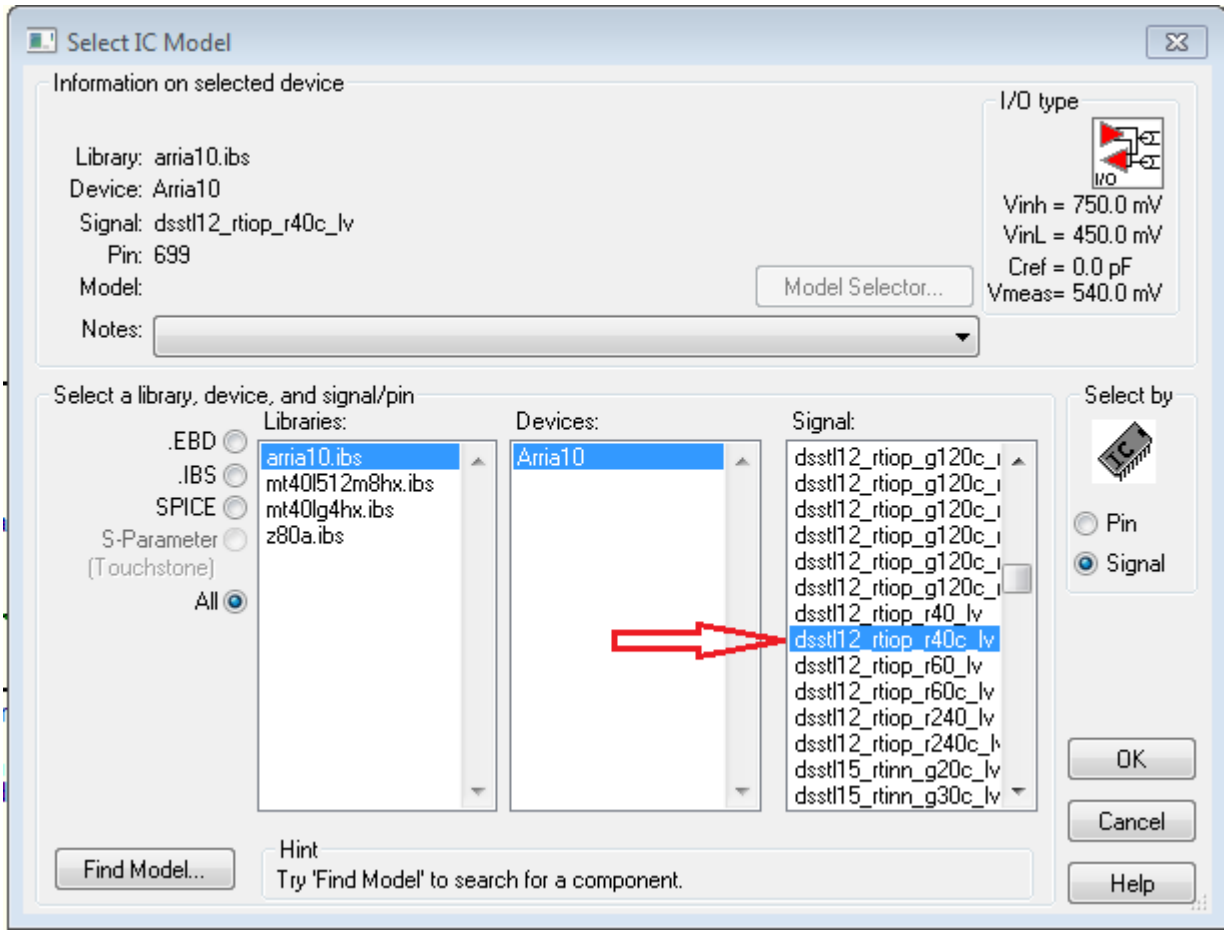
Chapter 5: Clock Simulation and Analysis

Clock simulation is very much like the C/A analysis except it is differential and very sensitive to Duty Cycle Distortion (DCD). Setting up the simulation is much complicated by the need to start with coupled differential signals with balanced length and using differential IBIS models. To this point we have the tools for most of this and the drivers are just named differently as disclosed in the IBIS file header. Because the 40 Ohm drivers worked well with 40 Ohm transmission lines it will be a good place to start.

Basic Clock Schematic

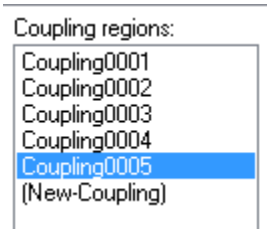
To place a differential IBIS part, select the differential driver symbol  . Then assign a differential driver to the symbol. When you add the differential model, the opposite polarity model is assigned to

the other output.



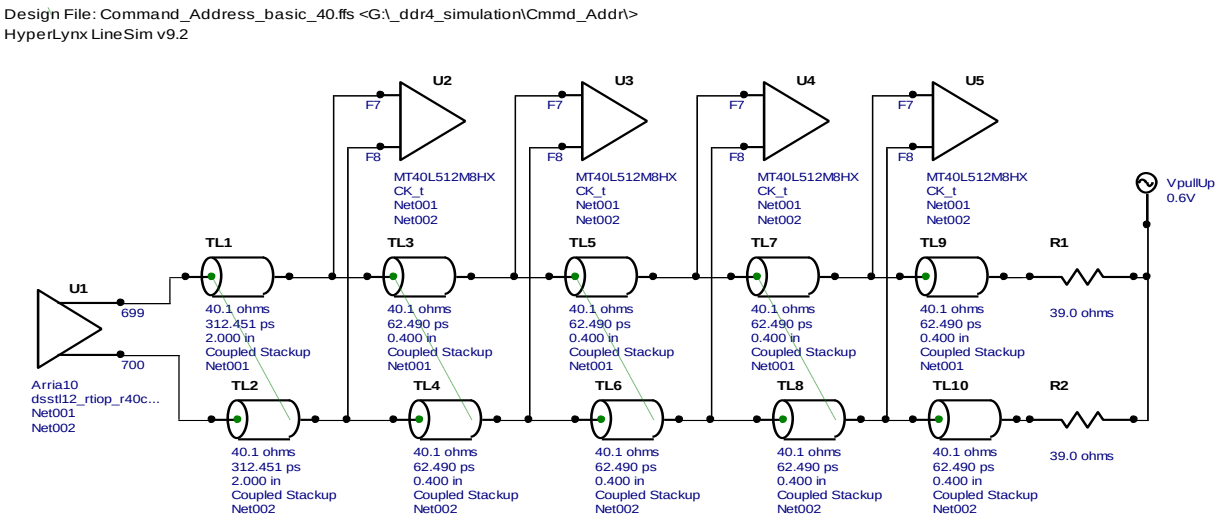
Once you have the Arria 10 model assigned, repeat the process for the memory devices by adding a differential symbol from the ribbon () and assigning the proper model from the memory IBIS file. Once you have one instantiated, copy it for the remainder of the memory parts so you do not have to repeat the assignment.

Place T-lines, adjust their lengths and couple the differential segments according to the board stackup plan. Be careful to give each segment its own coupling region.



Then add the termination to the topology.

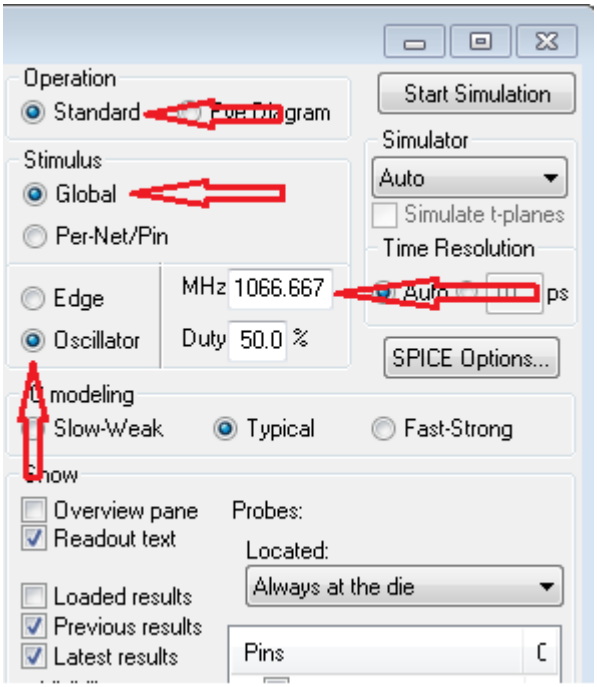
The finished schematic could look something like this:



Pre-Layout Clock Simulation

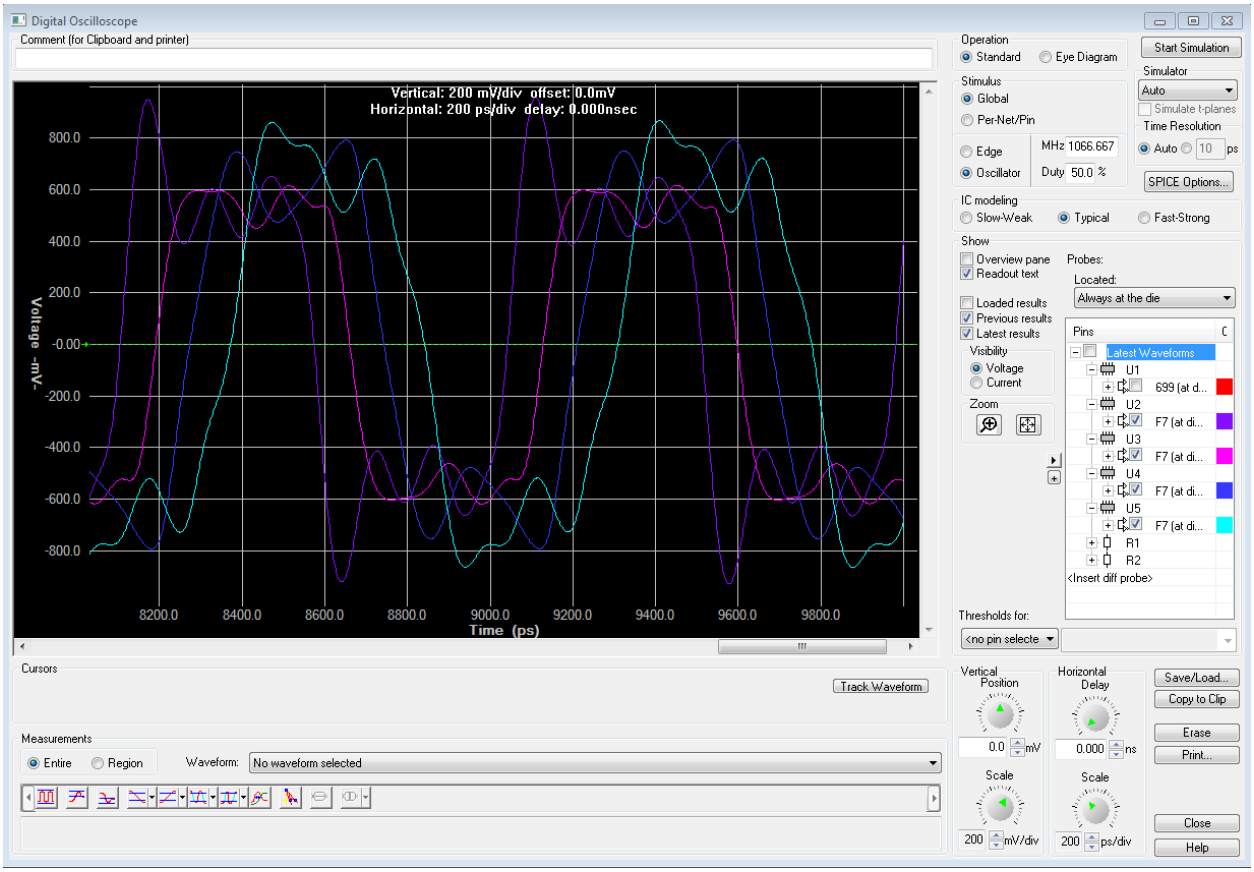
Since the clock is an oscillation the simulation setup is a little different.

1. Just open the interactive simulation window  .
2. Set up the simulation as Standard, Global, Oscillator, and 1066.667 MHz

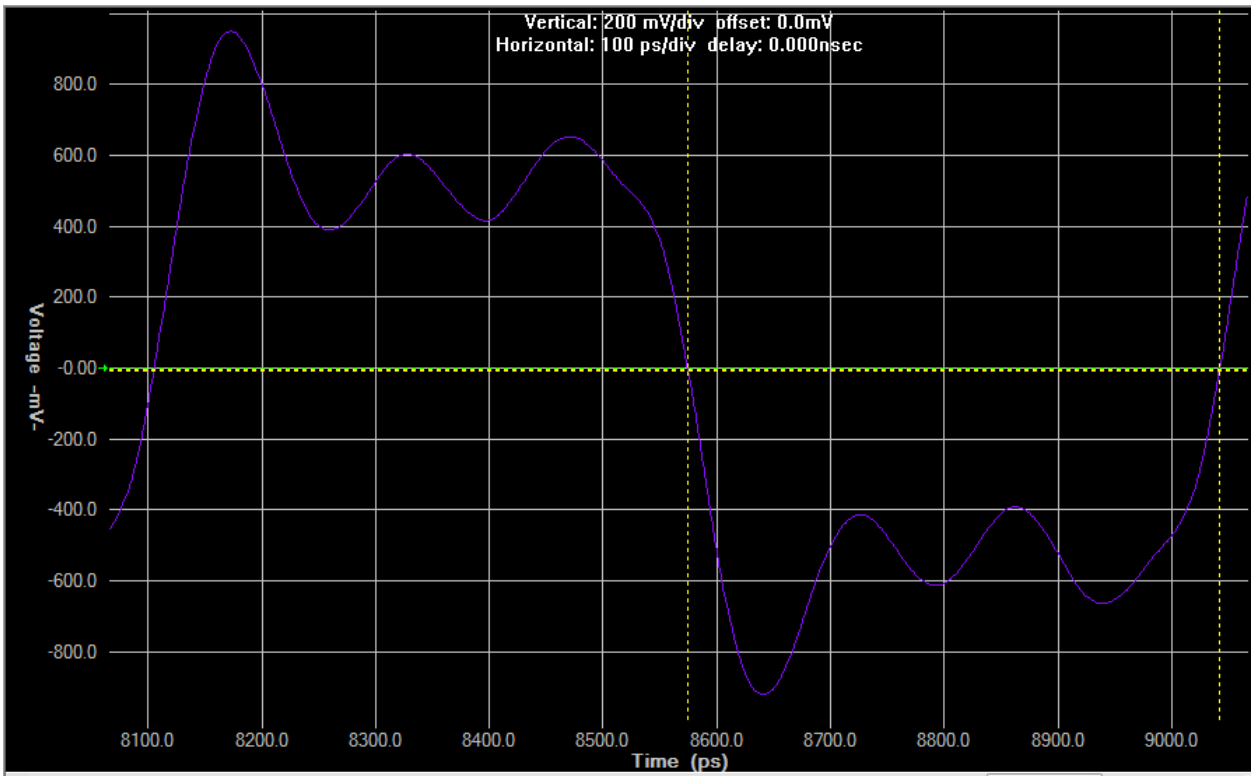


3. Run the simulation to see the waveform. If there are not at least 10 cycles of the signal, then increase the horizontal scale to obtain many cycles so that the DC balance is good. Then reduce the horizontal scale and use the position bar on the display to scroll to the right end to observe

the waveforms for the clock.



4. There are a couple of things that are of interest in these waveforms. One of them is the overshoot difference, especially with reference to the signal U2. The other is duty cycle distortion (DCD). The DDR4 JEDEC specification indicates that DCD is constrained to fairly small values. Turn off all the waveforms except U2 and analyze the zero crossing time for each half cycle.



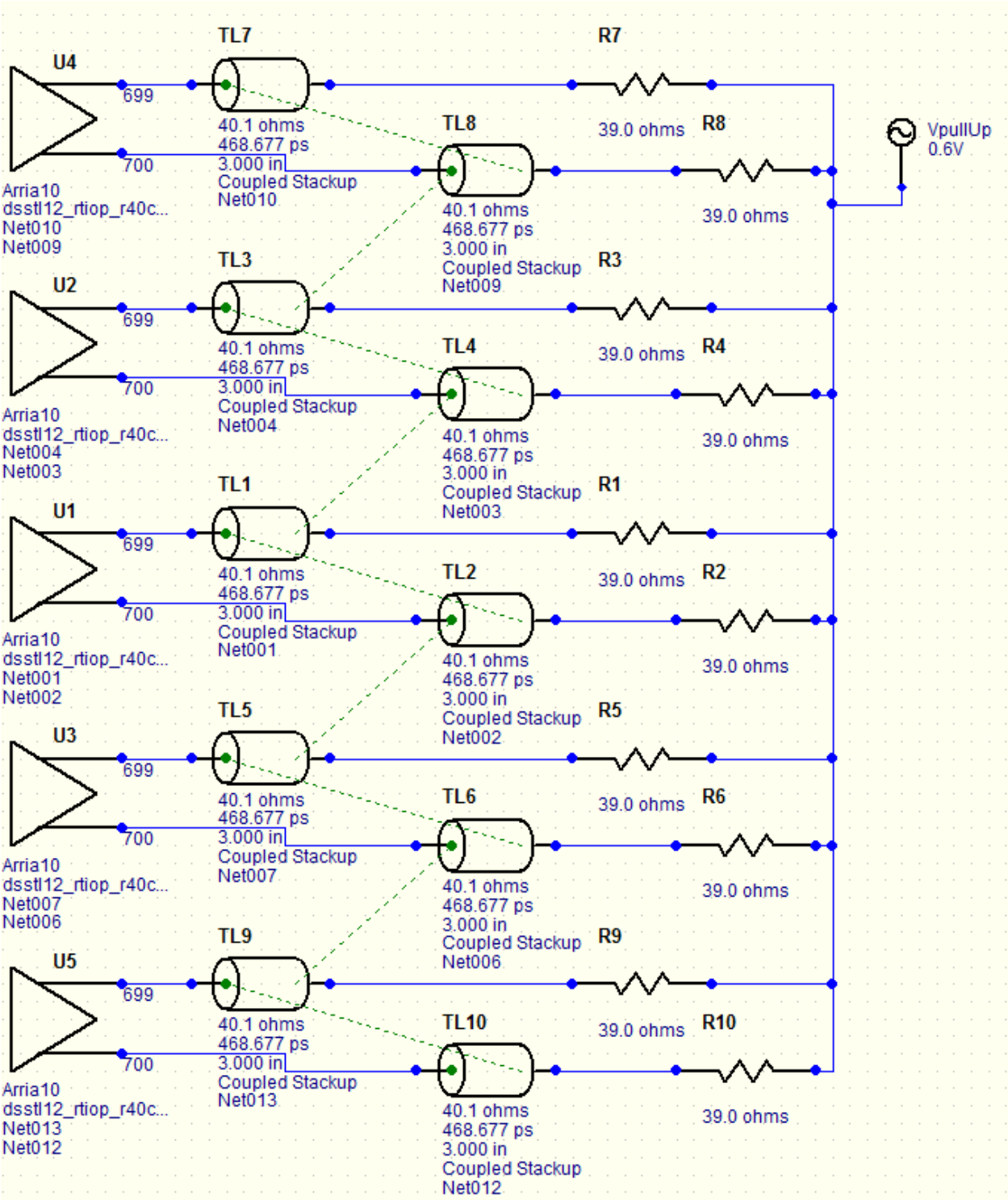
This turns out to be 468.6ps for the positive side and 467.4 for the negative side. This is not a big difference. The DCD is really low, so we are probably good to go with a 40 Ohm driver, 40 Ohm board impedance and 39 Ohm termination.

5. Other simulations can be created with different impedance driver/board/termination values to see what happens. This effort is and optional exercise for the designer to see what seems best.

Pre layout clock crosstalk

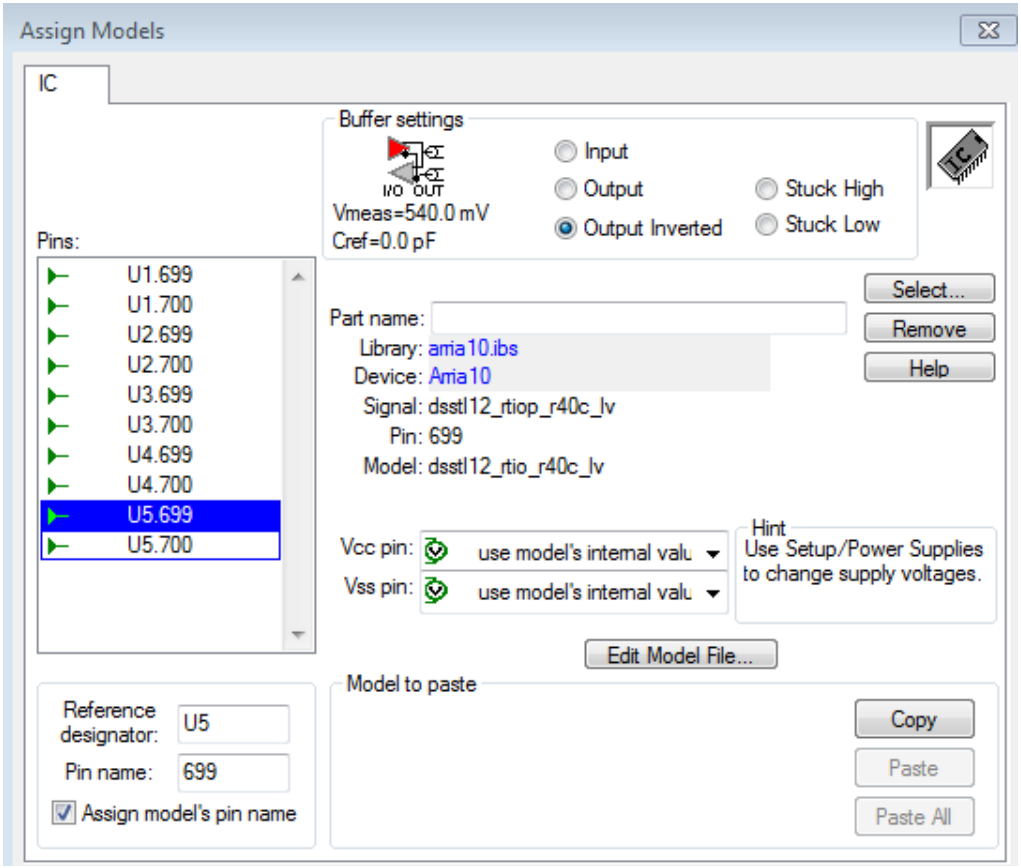
1. First, set up the topology using the same widths as were used for the single line clock simulation. Start with all of the T-lines uncoupled, as above, and go down through the

schematic, coupling the lines



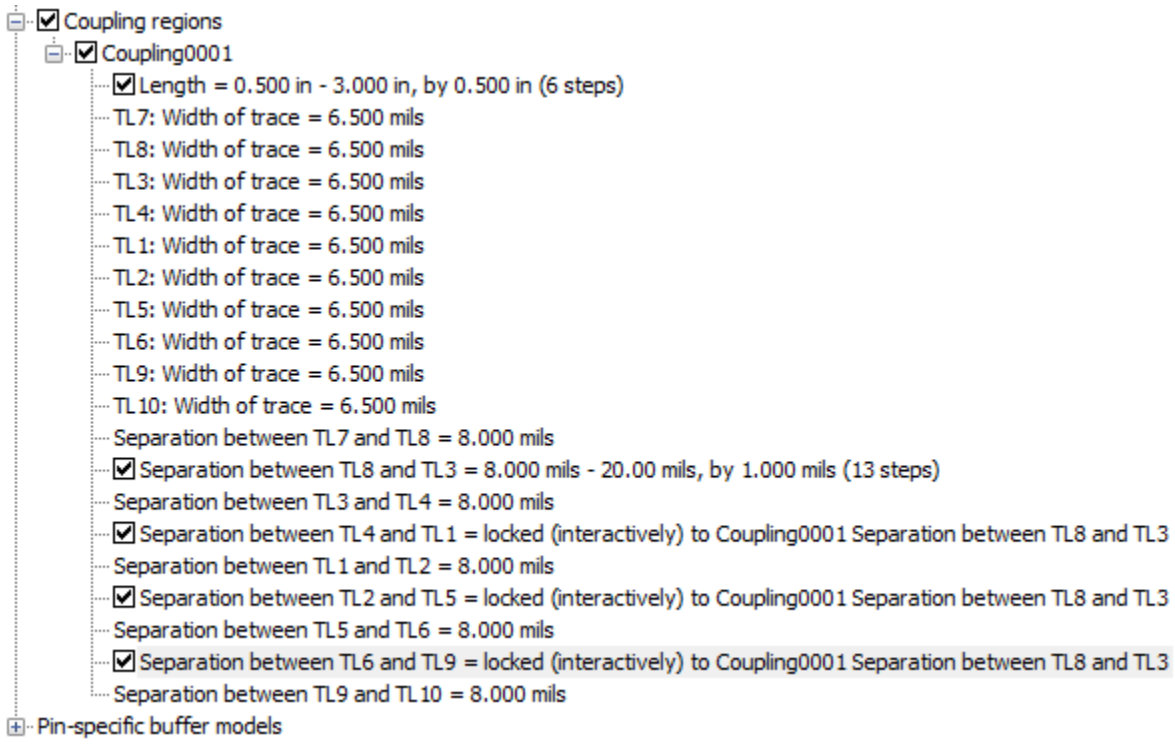
2. Configure the drivers by double clicking on any driver
    - The positive side of U1 to Stuck High
    - The positive sides of U2 and U4 to Output

- The positive sides of U3 and U5 to Output Inverted to cause the most coupling

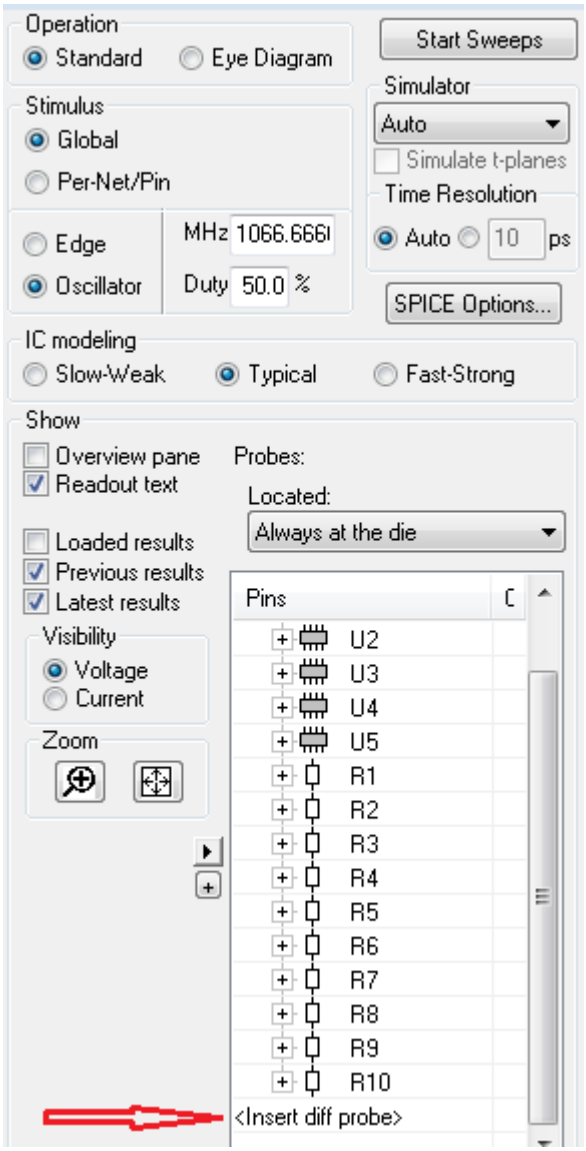


3. Set up the simulation for crosstalk by opening the interactive sweeps window 
4. Add ranges for the length and couple separation, use the “Copy Range” and “Paste Range as a Lock” buttons when doing more than one separation in sync. This limits the number of

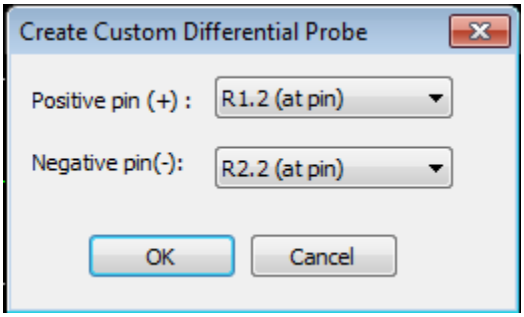
simulations that need to be run



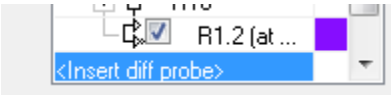
5. Click on Run Sweeps and set the oscilloscope up for Standard operation, Global Stimulus, 1066.666MHz Oscillator at 50% duty cycle, then double click on <Insert diff probe>



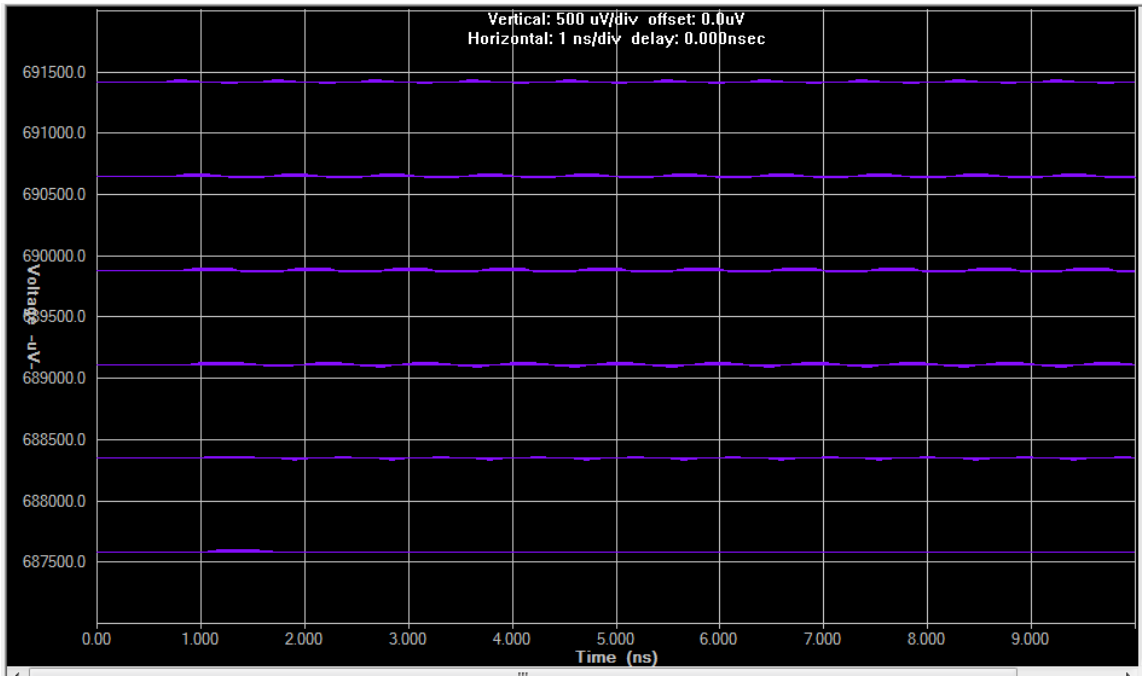
6. Select the two resistors for the victim channel, in the case of the schematic, above, these are R1 and R2 then click OK



7. The only signal that needs to be selected for the simulation is the one just added, the differential probe



8. Select Start Sweeps and zoom in on the results, notice that there seems to be very little coupling when starting with the nominal, 8mils, differential spacing



9. Click on Save/Load and save the results to a .csv file
10. Open the .csv file in a spreadsheet and analyze the crosstalk amounts
    - Use the same sort of formulas as shown for the C/A system analysis, above
    - Note that the maximum coupling is not very much, it is always less than -84db
    - This indicates that 8mils is probably going to be OK 84db is quite a lot of isolation

	A	B	C	D	E	F	G
1	HyperLynx Oscilloscope .CSV file						
2							
3	Design file: CLOCK_40_XTLK.FFS		Designer: David Lieby				
4	Created on: Wednesday, October 07, 2015		Time: 9:53:49 AM				
5	Max diff		2.91E-05	3.2E-05	3.27E-05	4.27E-05	2.96E-05
6	db coupling	-84.1573	-87.51699405	-86.6822	-86.4845	-84.1573	-87.3303
7		Time	Probe V V [R1.2 (at pin)/R2.	Probe V V	Probe V V	Probe V V	Probe V V
8		0.00E+00	0.6914212	0.69065	0.689881	0.689114	0.688348

Chapter 6: DQ/DQS Channel Simulation and Analysis

The drive standard for the DQ/DQS system for DDR4 is specified to be Pseudo Open Drain (POD). In POD the signal lines are terminated by an impedance to Vcc and not a synthesized Vtt for the system. The reason for this is the reduction of simultaneous switching noise for the system. The impact that this

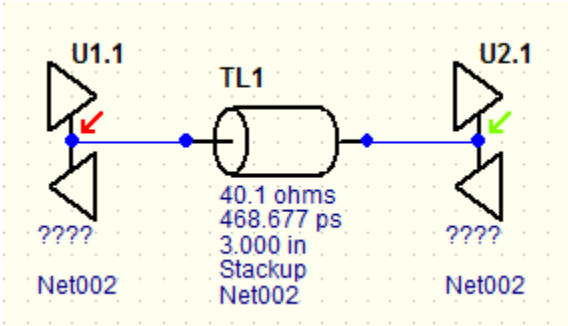
standard has on the system is profound. With no Vtt constantly establishing a nominal threshold for the eye diagram things get a little fuzzy when it comes to determining the best driver/board/termination values for a particular system.

Termination

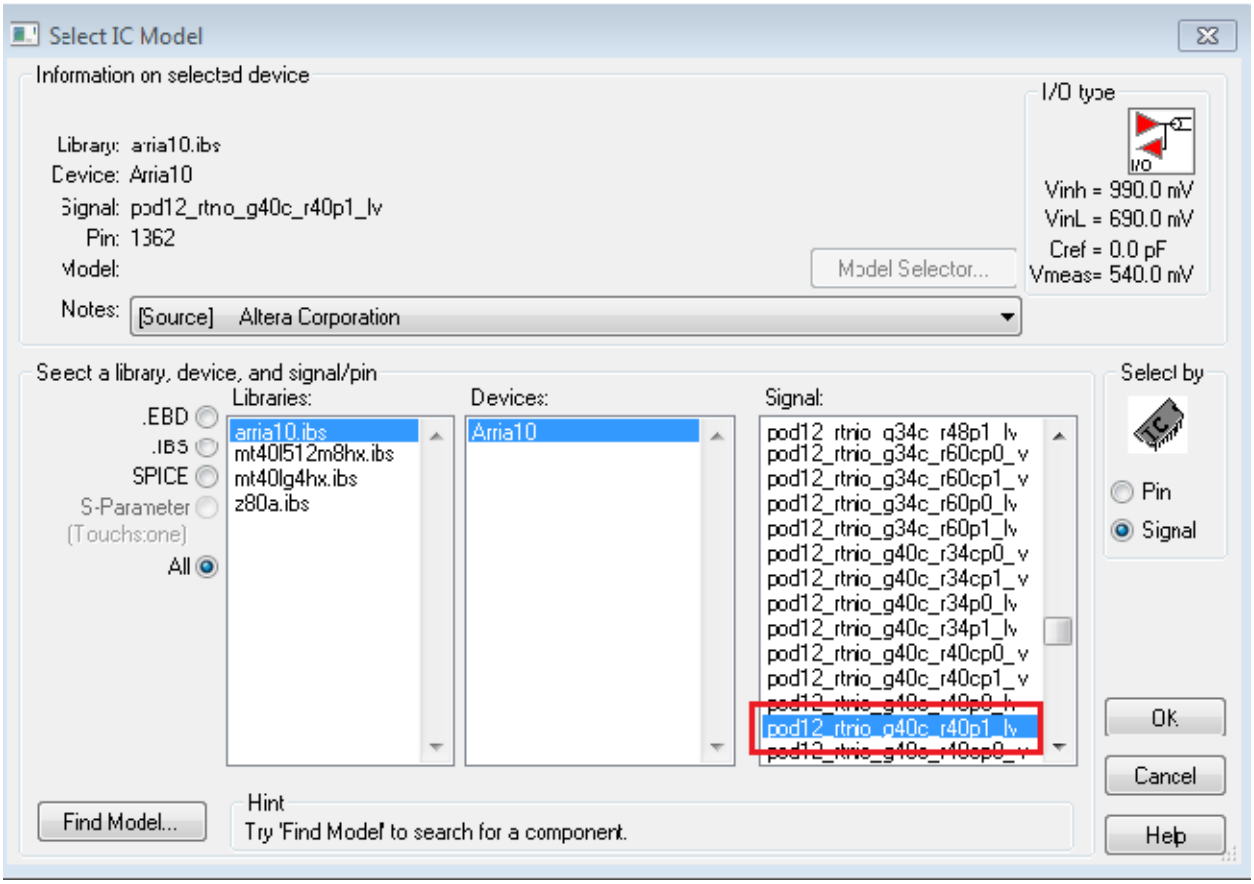
The nominal impedance for the DDR4 has been set to 40 Ohms during the C/A simulations. A DQ topology, below, was created to determine what the optimal impedance would be for the drivers and receivers.

Initial setup

Create a single ended topology with the same T-line as the CA system with a single ended driver on each end using the single line model placement on the top ribbon. 



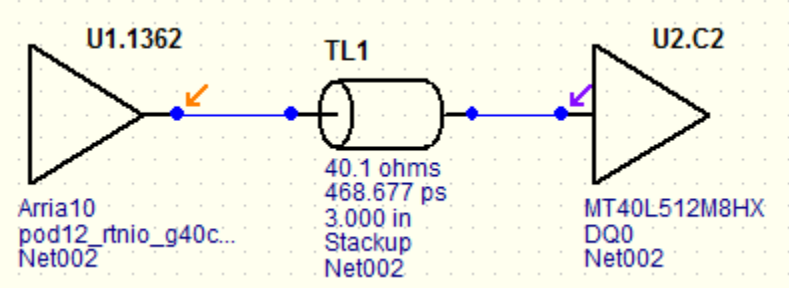
Selecting the driver for the Arria 10 is fairly straight forward. Double click on the symbol for U1.1 and select a POD symbol with 40 ohm drive and termination



and click OK.

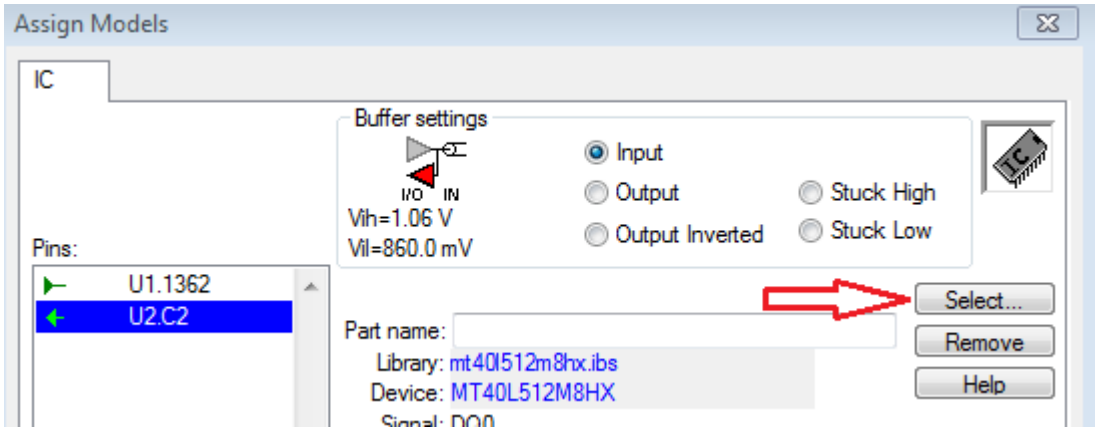
Repeat the process for U2 by selecting DQ0 from the device.

Nominal Topology to the Memory Device

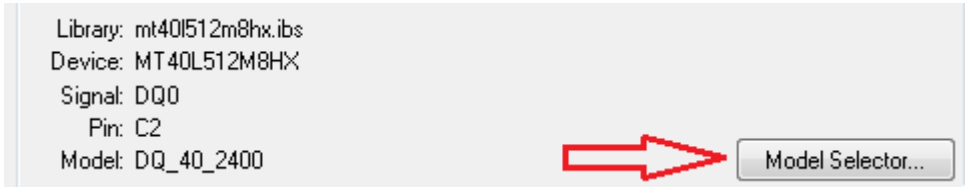


The memory device IBIS file was created to use sub-models and selecting them is done using the model selector. The user should also be familiar with how the IBIS file provider meant for the models to be used. The Micron models provide drive termination separate from receive termination to keep the models clean so the simulation requires changing the sub-model and not just changing it from input to output.

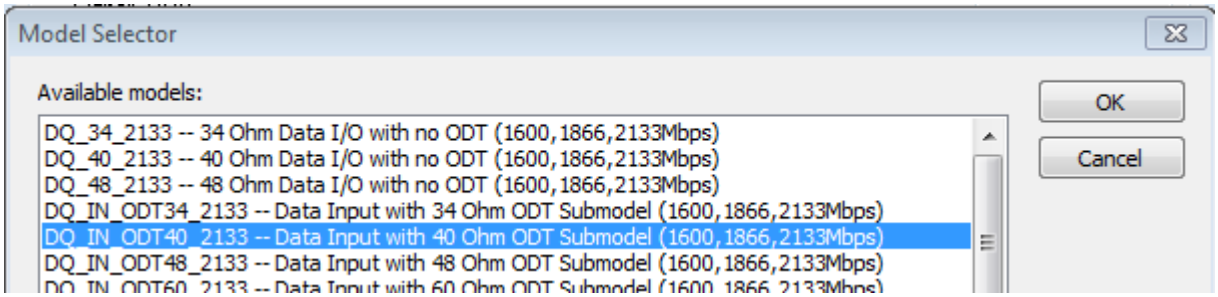
1. Double click on U2 to bring up the Assign Models pop-up then click on Select to bring up the Select IC Model pop-up



2. Click on the Model Selector to discover the options



3. Select the model needed

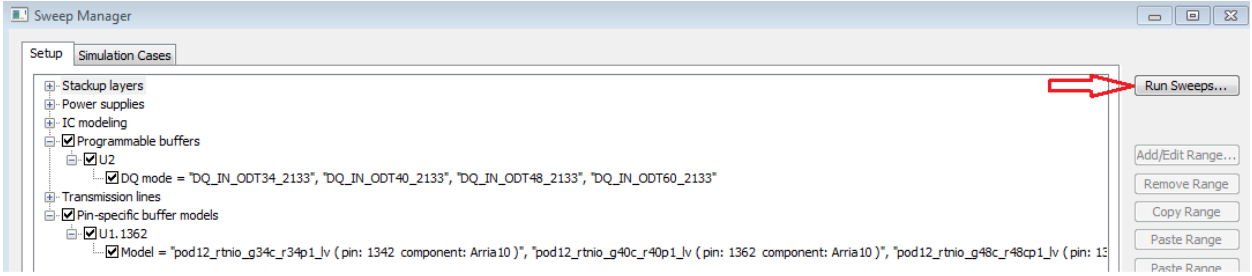


then click OK up through the pop-ups.

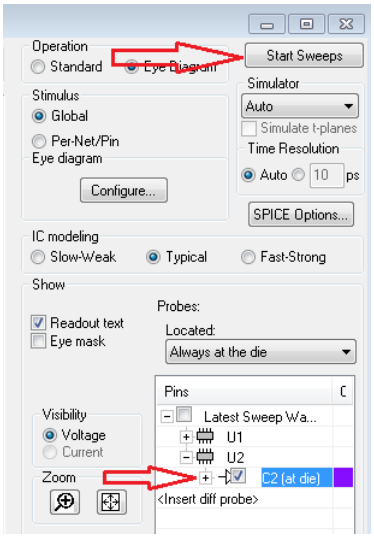
Setting up and analyzing ISI termination simulations for DQ signals

Open up the interactive sweep manager by clicking on the icon -  - or using the dropdown: Simulate SI -> Run Interactive Sweeps

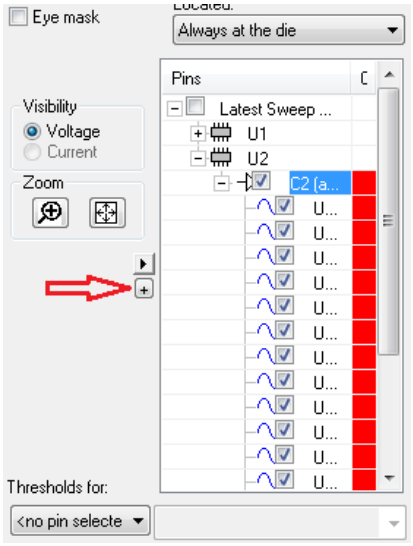
1. FPGA Drive strengths of 34, 40, 48, and 60 Ohms and terminations of 34, 40, 48, and 60 Ohms were selected for the memories. The impedance of the T-line is still 40. Select Run Sweeps to get to the Oscilloscope window.



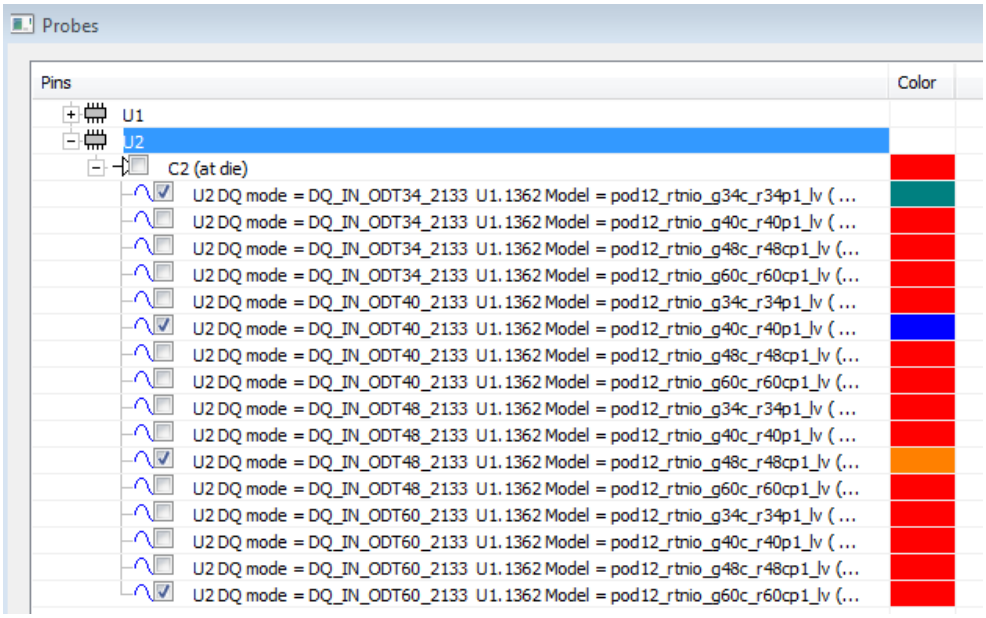
2. Set the simulation up for an eye diagram at 1066.667MHz and select U2 for analysis and click “Start Sweeps” to get the job done.

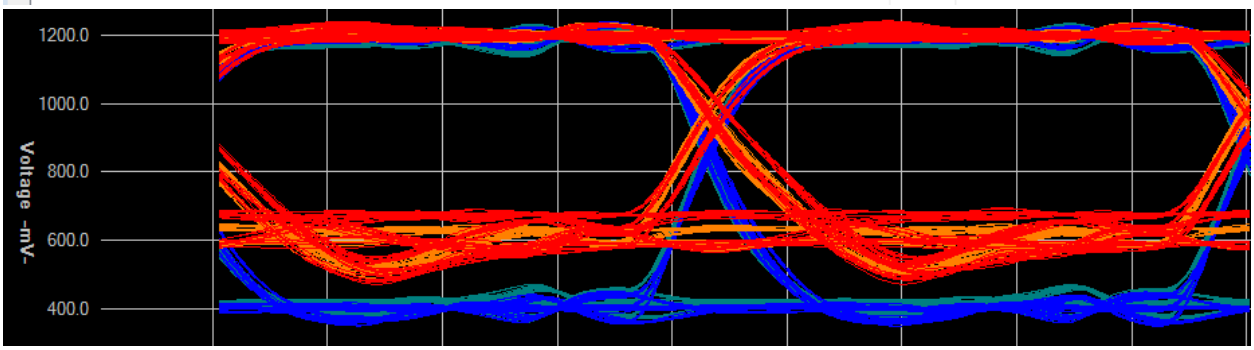


3. When the simulation is complete, open the probes pop-up by clicking on the + indicated below

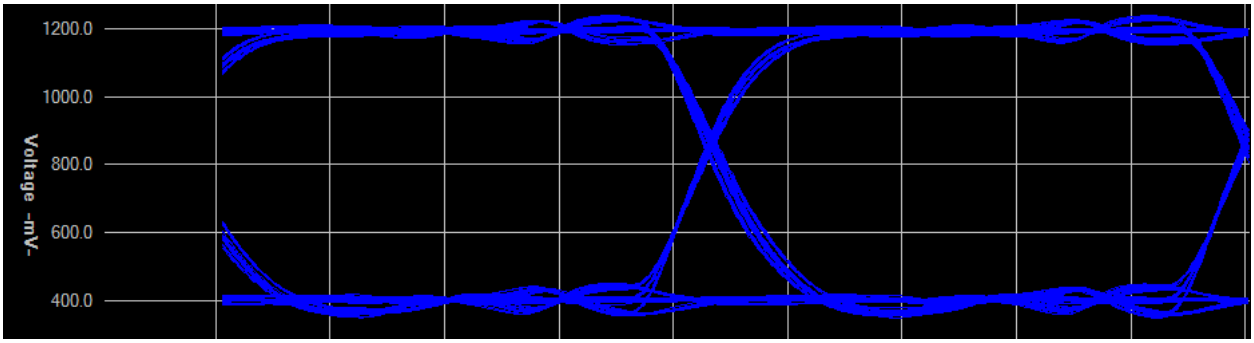


4. You can expand the pop-up width to see the characteristics of the drivers and receivers, and make changes to the color of each to compare the waveforms





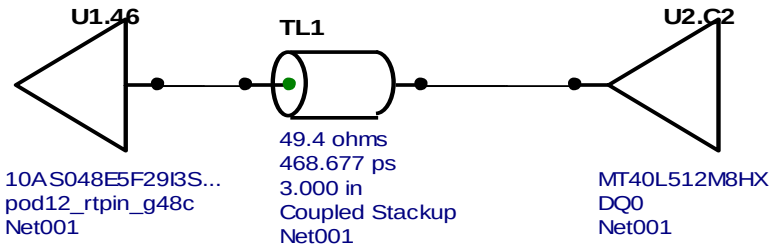
5. The best eye is given when the driver and termination match the T-line impedance



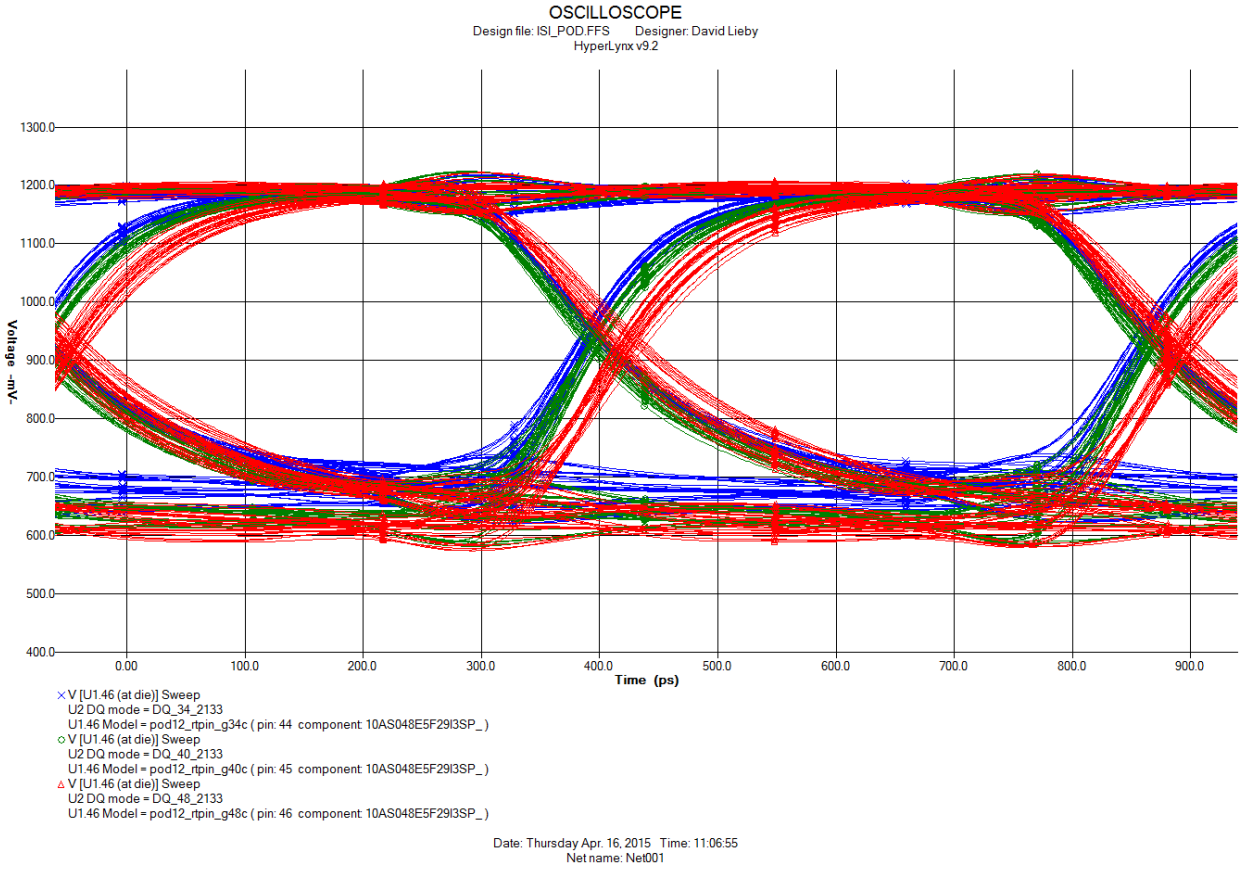
6. If you use imbalanced drivers and terminations to achieve an eye with more vertical height it will not help in the long run. One must be careful not to introduce excessive overshoot or deterministic jitter around the crossing point.

Nominal Topology from the Memory Device

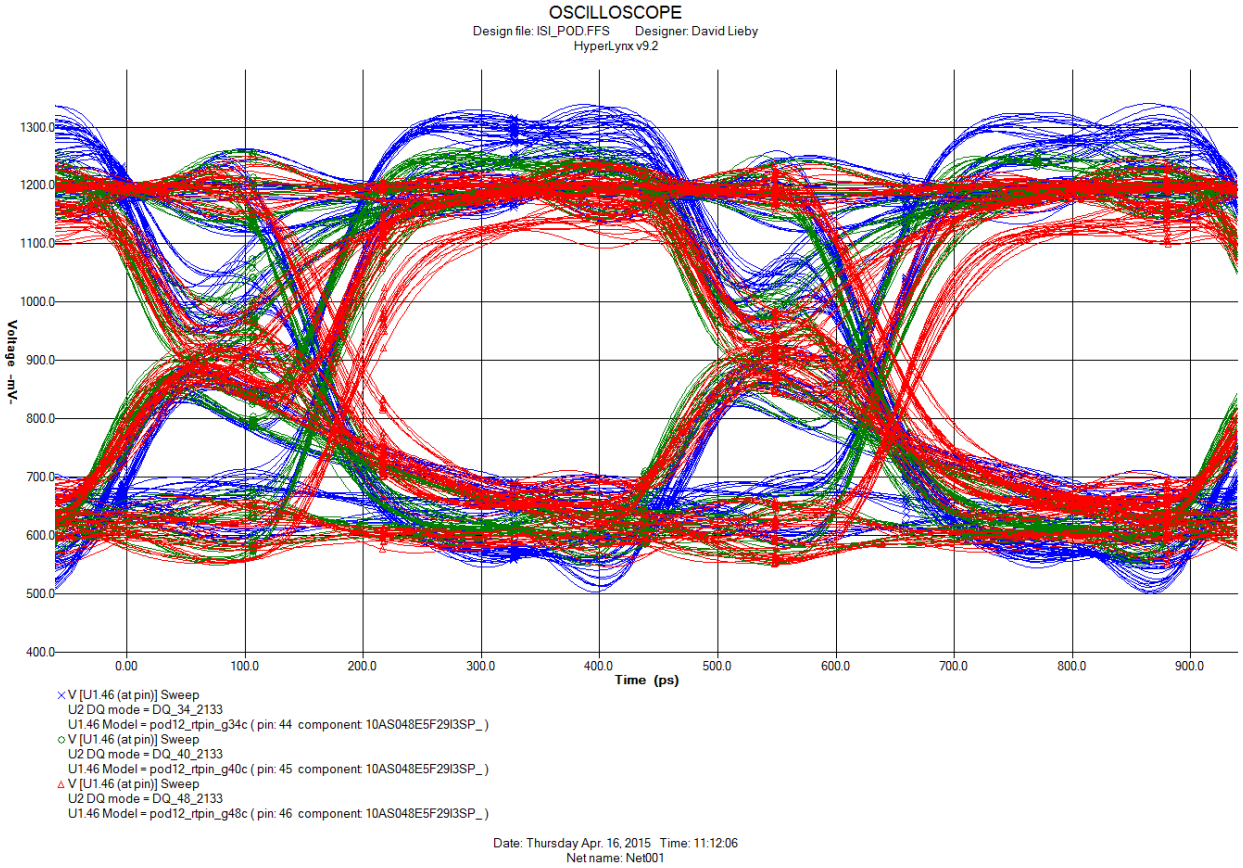
Design File: isi_pod.ffs <C:_AN_Docs\DDR4_sim_guide\>
HyperLynx LineSim v9.2



Running a set of sweeps as was done above for the transmit eye the following waveforms were observed at the die.

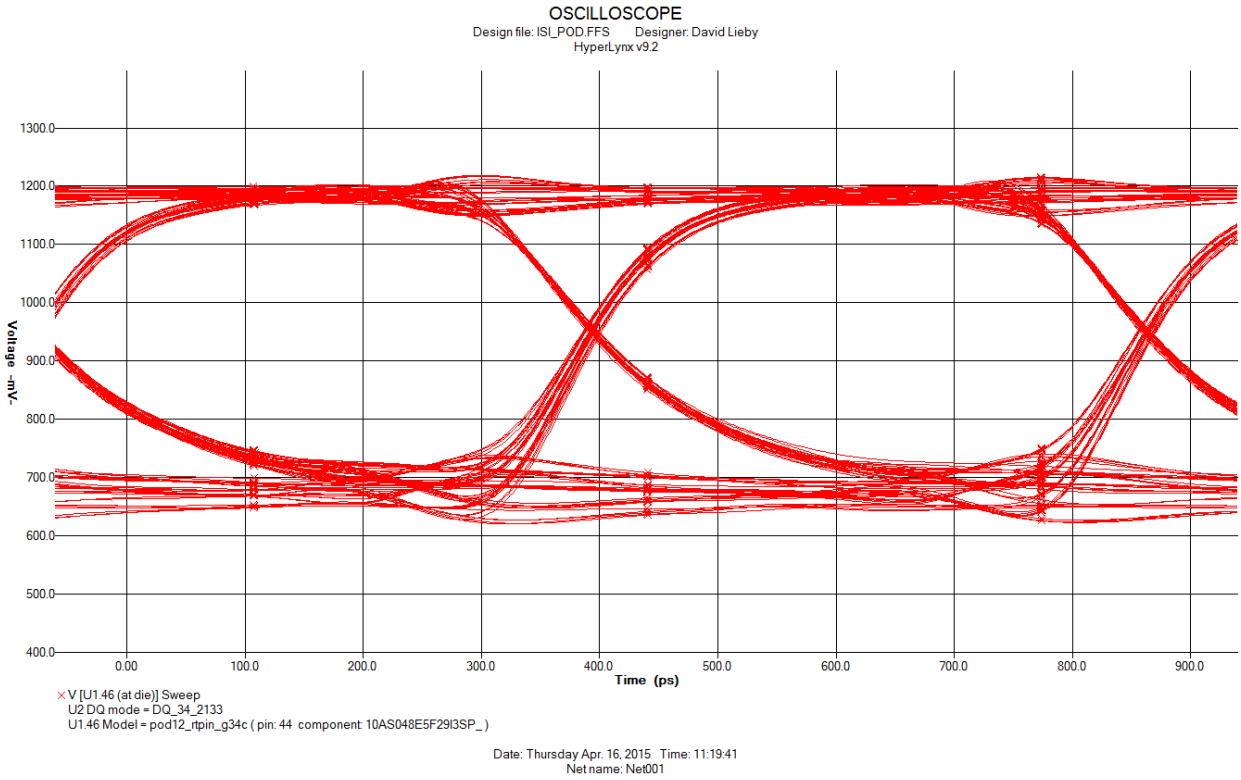


The same simulation created with the observation point at the pin of the FPGA looks like this.



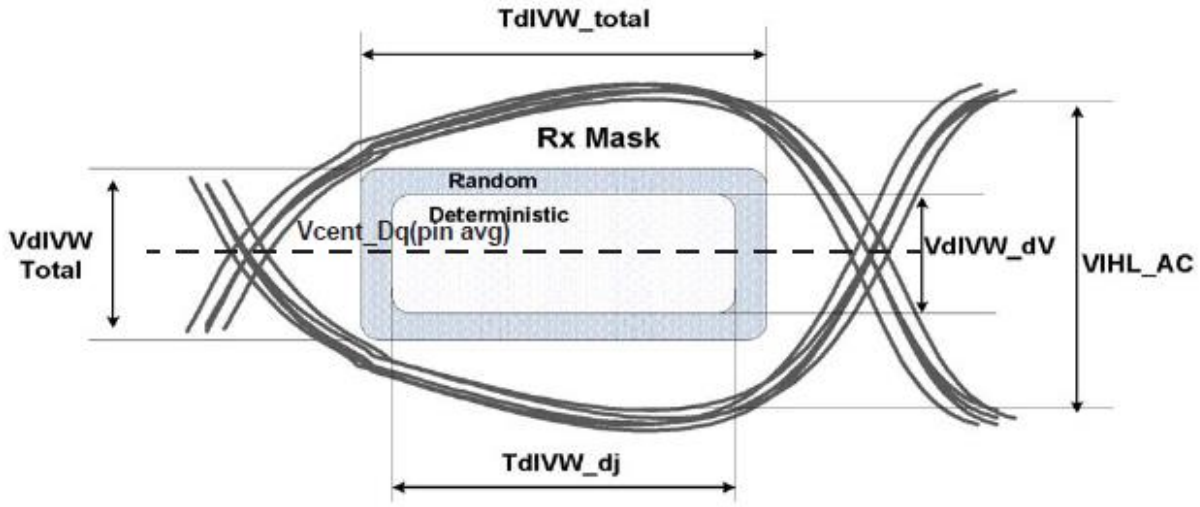
This illustrates the apparent RLC interference with the signal when observed with an oscilloscope. This simulation result should be provided in a signal integrity report for the interface so the engineers in the lab will not be confused with the measurements they may make.

The best waveform set for the receive system was one that used 34 Ohm for the ODT and OCT values as seen here.



Transmit Eye Mask Opening

The eye mask for the DQ/DQS systems in DDR4 is determined differently from previous generations of DDR. For the eye opening height the number to use is Vcent_dq +/- the threshold value determined by the data rate of the interface.



For data rates up to 2133Mbps the value is +/- 68mv. For data at 2400Mbps it is +/-65mv and above that is +/-62.5mv.

The training that must take place during initialization for any DDR4 implementation establishes the best level for the internal threshold voltage for the entire chip. As a result of this training the best possible eye should resul0t.

Receive Eye Mask Opening

The process used for determining the best reverence voltage on the FPGA side should be identical to the one used to set the DRAM threshold. The eye opening height is +/-70mv from that value.

Crosstalk Constraint Simulations

Crosstalk is the coupling between two or more conductors running close to each other. Some teams use the term “Simultaneous Switching Noise” (SSN) for this but here we will use crosstalk to differentiate it from the effects on the signal due to power supply collapse normally indicated by SSN. The amount of crosstalk is determined by several factors. On a printed circuit board with controlled impedance traces, the coupling variables are the spacing between the traces and the length of the coupling. There will be more than one set of constraints, one for microstrip traces and another for the stripline traces.

There may be more than one layer of stripline traces in the case where a dual stripline stackup is involved. If the thickness and of the board is not as much an issue as the isolation between signals then single stripline an excellent solution. In the case of a PCIe form factor board that can only be 0.062” thick, then dual stripline will be necessary.

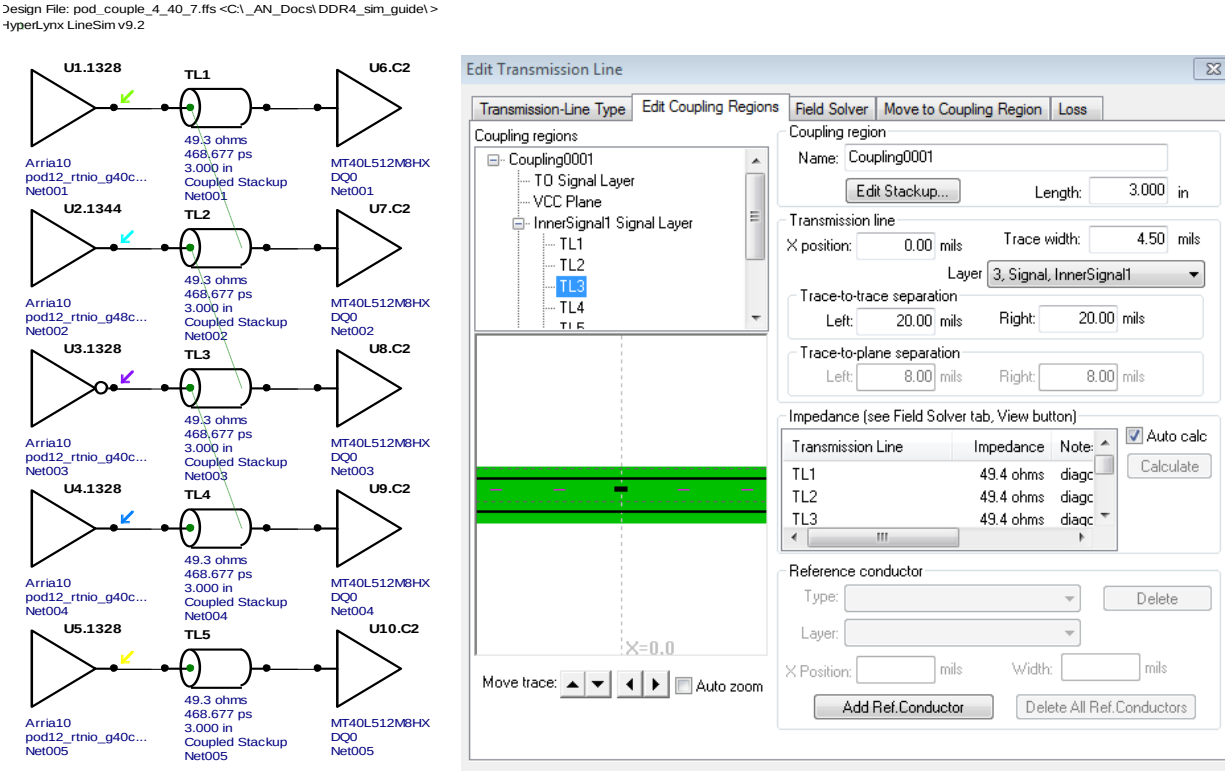
Example of stripline crosstalk simulation for parallelism rules

Before the board is laid out a set of parallelism rules must be established. These rules are established to guarantee that the signal will not be distorted beyond certain limits by nearby signals. Below is a description of how to use HyperLynx to determine parallelism spacing rules.

A crosstalk topology

The topology is created on one layer of a dual-stripline layer with one victim (U3->U8) and four aggressors. The simulation is run with the output of U3 stuck high and all of the aggressors toggling at once and in phase to yield the worst-case crosstalk. These should be executed using the drive strength and termination found above for eye optimization.

This simulation is for a stripline layer. The simulation topology for a microstrip layer is similar.



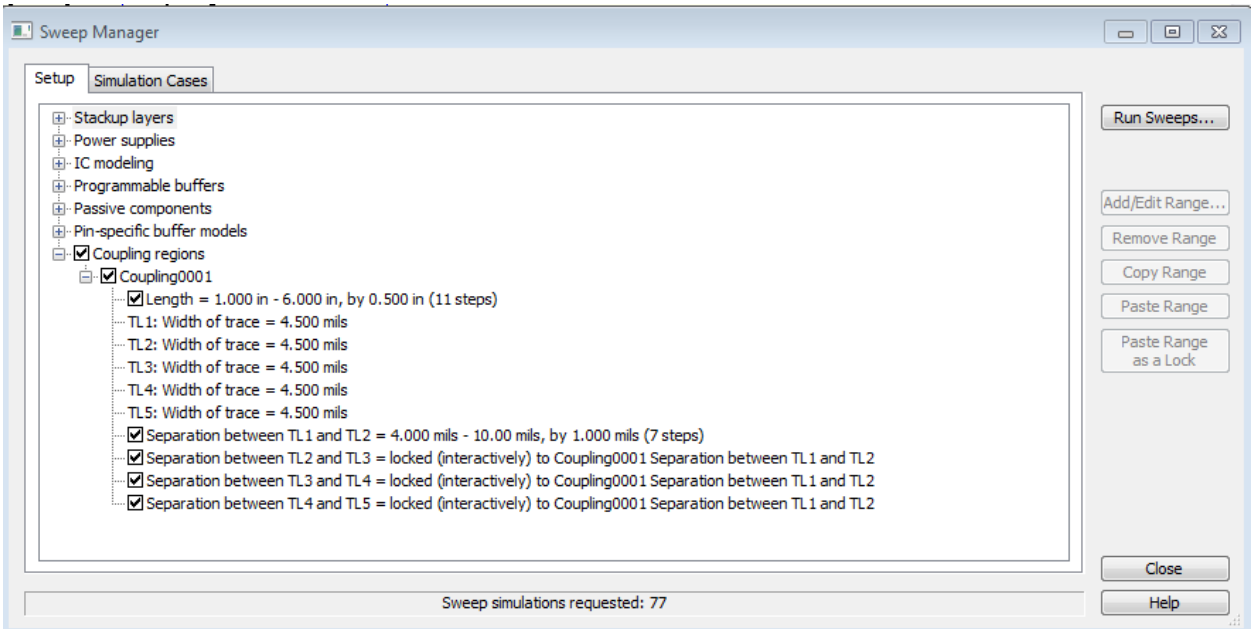
Setting up the simulations

Newer versions of HyperLynx can run interactive sweeps of various properties in the topology.

Open up the interactive sweep manager by clicking on the icon -  - or using the dropdown: Simulate SI -> Run Interactive Sweeps

For this system we will sweep:

- The length of the T-lines from 1 inch to 6 inches in 0.5 inch increments
- The spacing between the T-lines from 4mils to 10mils



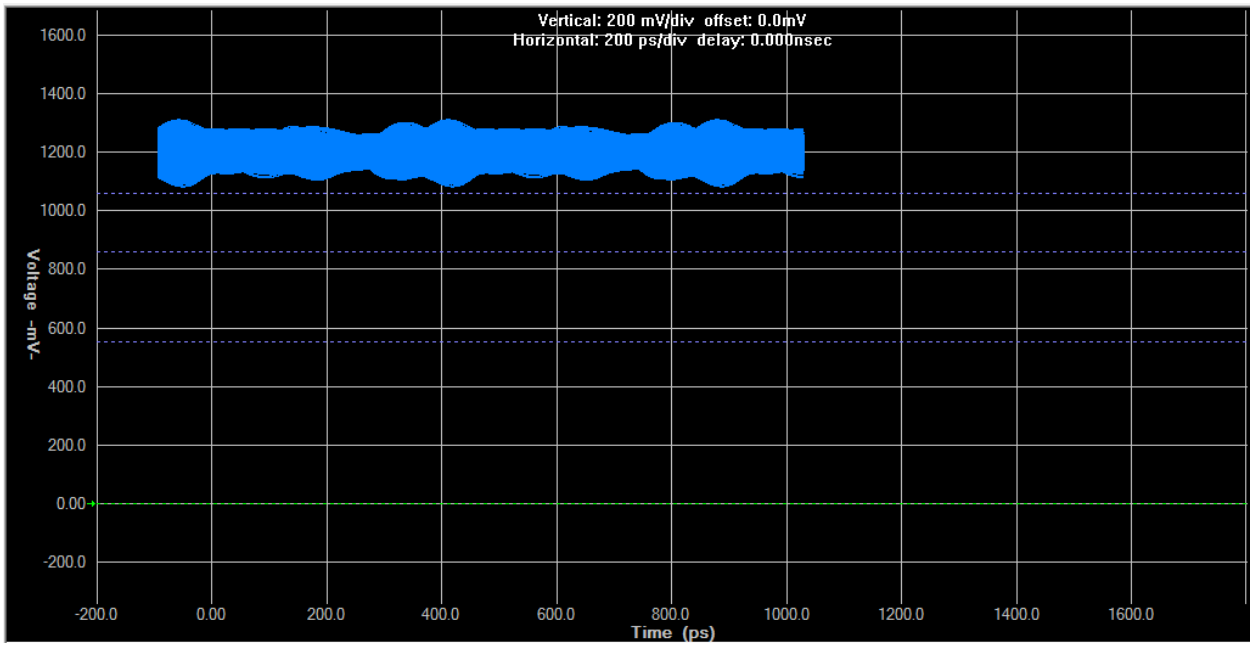
The simulation was run at 1066.666MHz for this system. The results for a faster or slower clock would not matter much as the coupling is related to the risetime of the signal and not the repetition rate.

You may need to adjust the sweep parameters to suit the needs of your system.

Running the Simulations

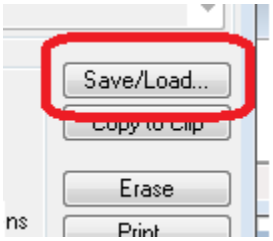
Clicking on the “Run Sweeps” button takes you to the Digital Oscilloscope where the simulations are executed. After the eye diagram simulation is set up and the latest sweeps are all checked, turn off the display of all the traces except the input to the victim signal, in this case U8.

The results of this simulation are shown here.



This looks very messy for a good reason. The beginning spacing of the lines for some cases is very close and the coupling is fairly high for those cases, also the propagation time for each length is different.

Next, export the waveform to a comma separated variable (.csv) file using the “Save/Load” button.



When the .csv file is opened with a spreadsheet program, formulas can be written to extract the peak to peak noise contribution for each set of spacing and lengths. The amount of crosstalk that can be tolerated should be determined by the system architect. The crosstalk will have an effect on the eye opening so a simulation should be created for the topology under consideration to determine the sensitivity of the channel. A simulation, below, will show the effects of the crosstalk.

Analysis of crosstalk for parallelism rules

When the “Saved” file is opened in a spreadsheet program the coupling values can be readily determined. The data is used by finding the minimum spacing for given coupling lengths for a given amount of crosstalk. The crosstalk in decibels can be calculated by finding the peak to peak maximum for each case and using the nominal voltage swing for the POD 1.2V interface for the considered drivers. Each designer has an idea about what a tolerable amount of coupling is. For this instance a 40 Ohm driver was used and the nominal eye height was determined to be 950mv.

DQ/DQS Simulation for ISI and Crosstalk Effects

Above the amount of crosstalk for a given parallelism was determined. If the engineer has decided that a -20db effect is sufficient, in the spreadsheet, above, the spacing was 7mils. It remains to see how that interferes with the system. First the disturbance caused by successive symbols on the eye diagram (ISI) is analyzed and then the effects of crosstalk added to see if the resulting eye diagram satisfies the system design goals. For more data on crosstalk please see the documentation at http://www.alterawiki.com/wiki/Arria_10_EMIF_Simulation_Guidance

ISI

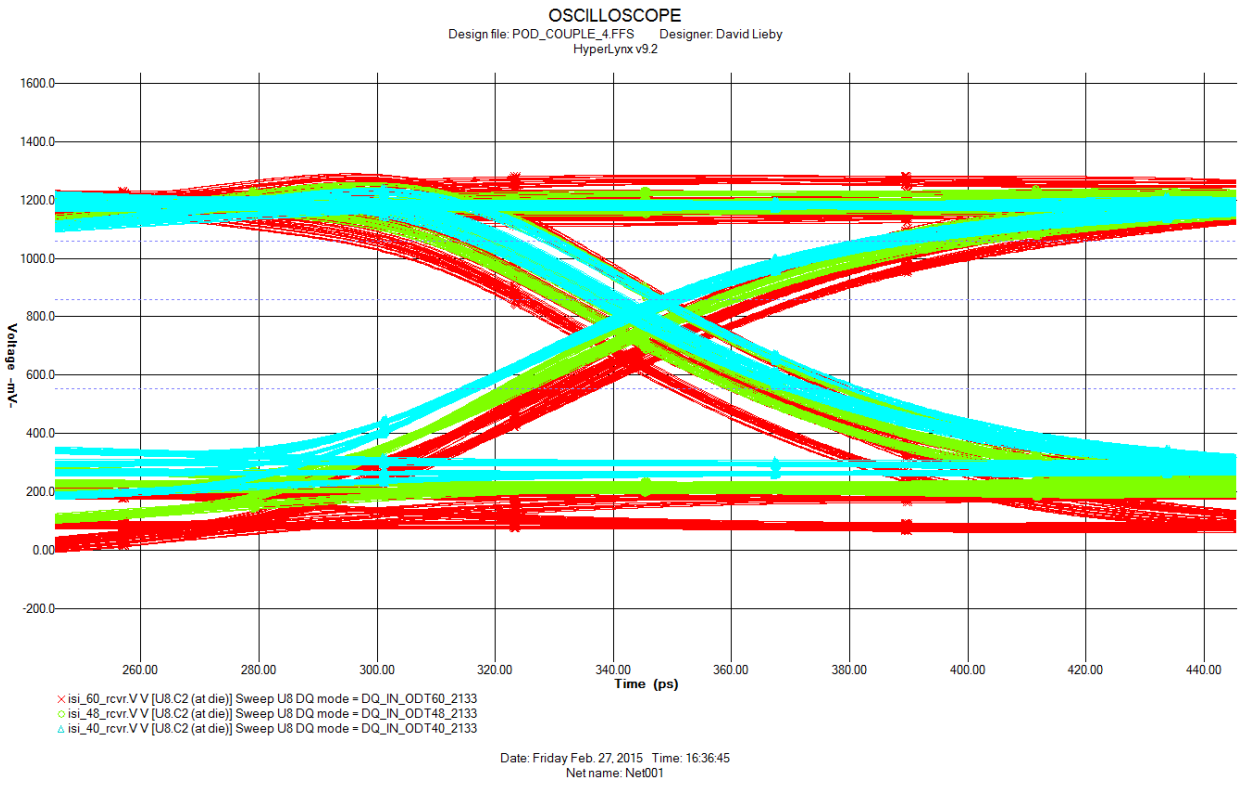
Inter Symbol Interference (ISI) is the interference between successive signals on a channel. Given a good driver, a channel with smooth impedance, and a termination that matches the transmission line, the signal should show no ISI. In the real world there are many things in the channel that are not matched to the board characteristic impedance. In addition the termination resistor value being off can cause a huge amount of interference. . For more data on ISI distortion please see the documentation at http://www.alterawiki.com/wiki/Arria_10_EMIF_Simulation_Guidance

Stackup and Topology Setup in HyperLynx

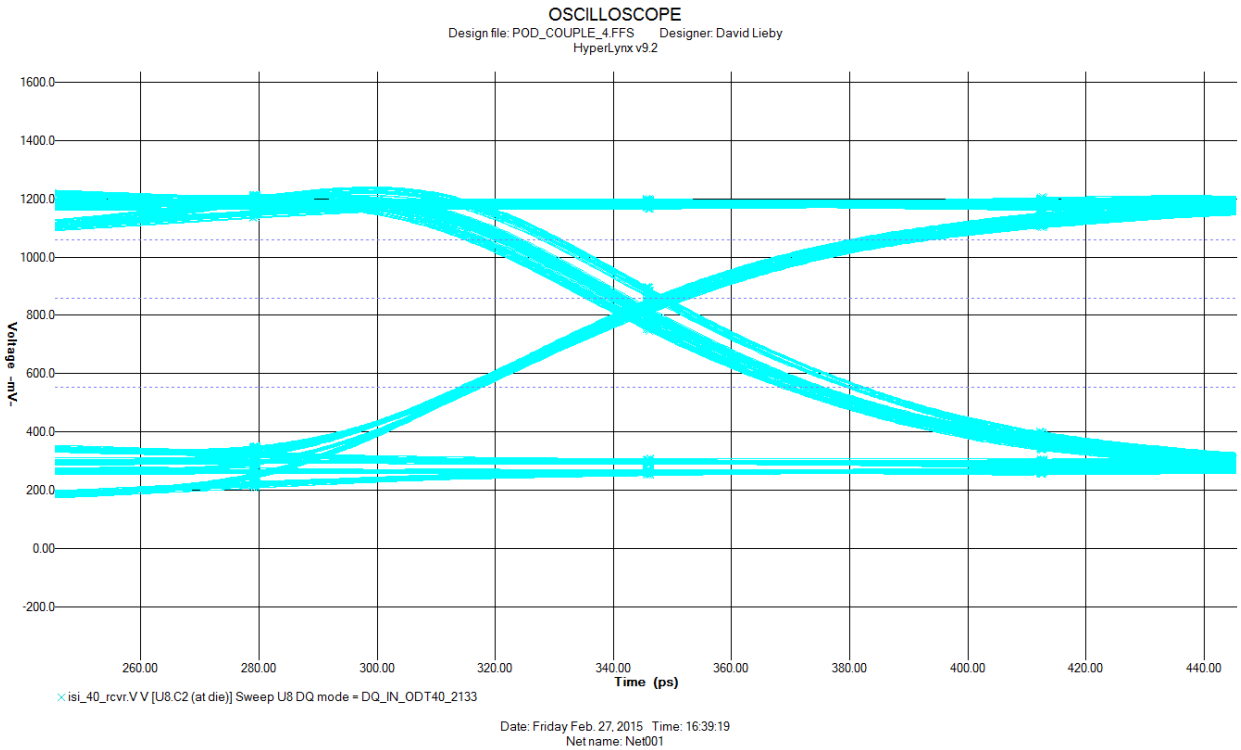
The stackup and topology are the same as those for crosstalk, above.

Termination variation simulation

Here we vary the ODT termination value between 40 Ohms, 48 Ohms and 60 Ohms with a 48 Ohm driver. Only the topology for U3 to U8 is used and the line spacing set to 20mils to eliminate crosstalk effects.



Here, when examined closely, it is evident that the 40 Ohm ODT receiver has the least amount of jitter at the crossing.



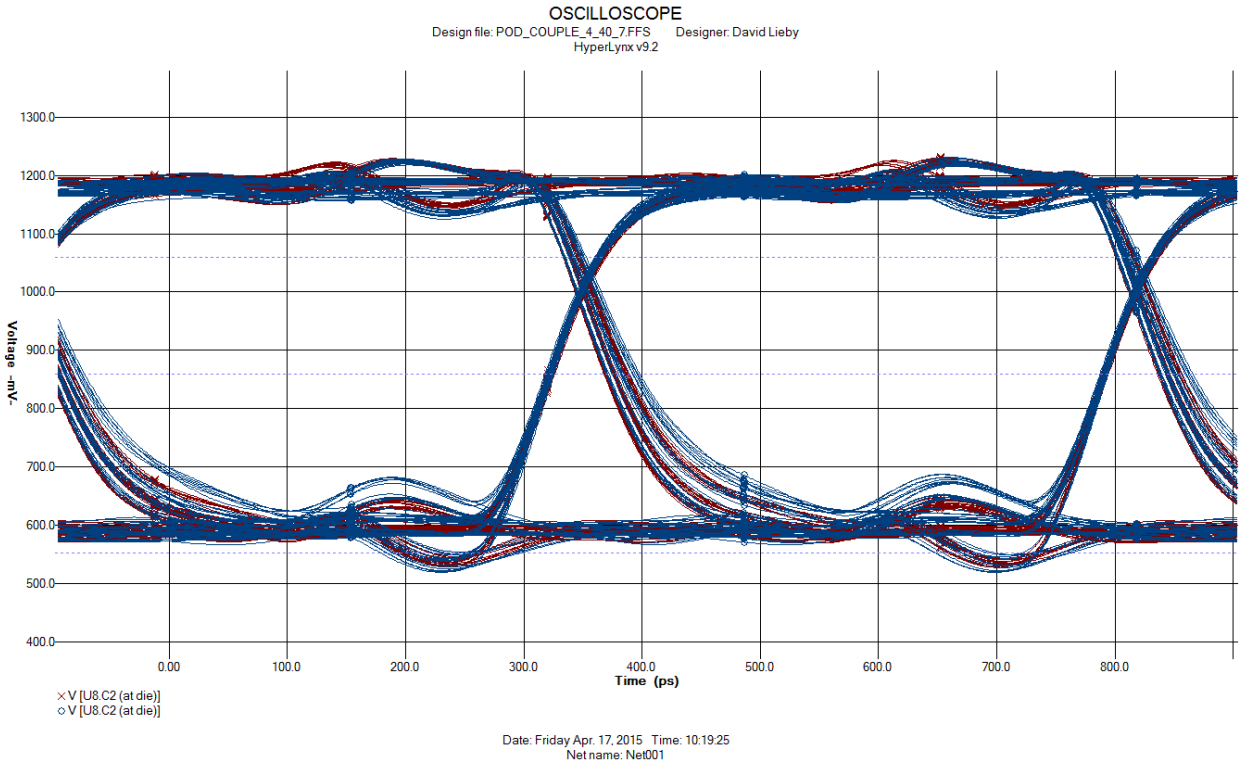
Because DDR4 has a calibration mode that must be used to set the threshold value for the individual part, this should be the best possible eye.

Other topologies can be simulated using these techniques. Please refer to the last section of this document titled “Further Reading” for references to additional material.

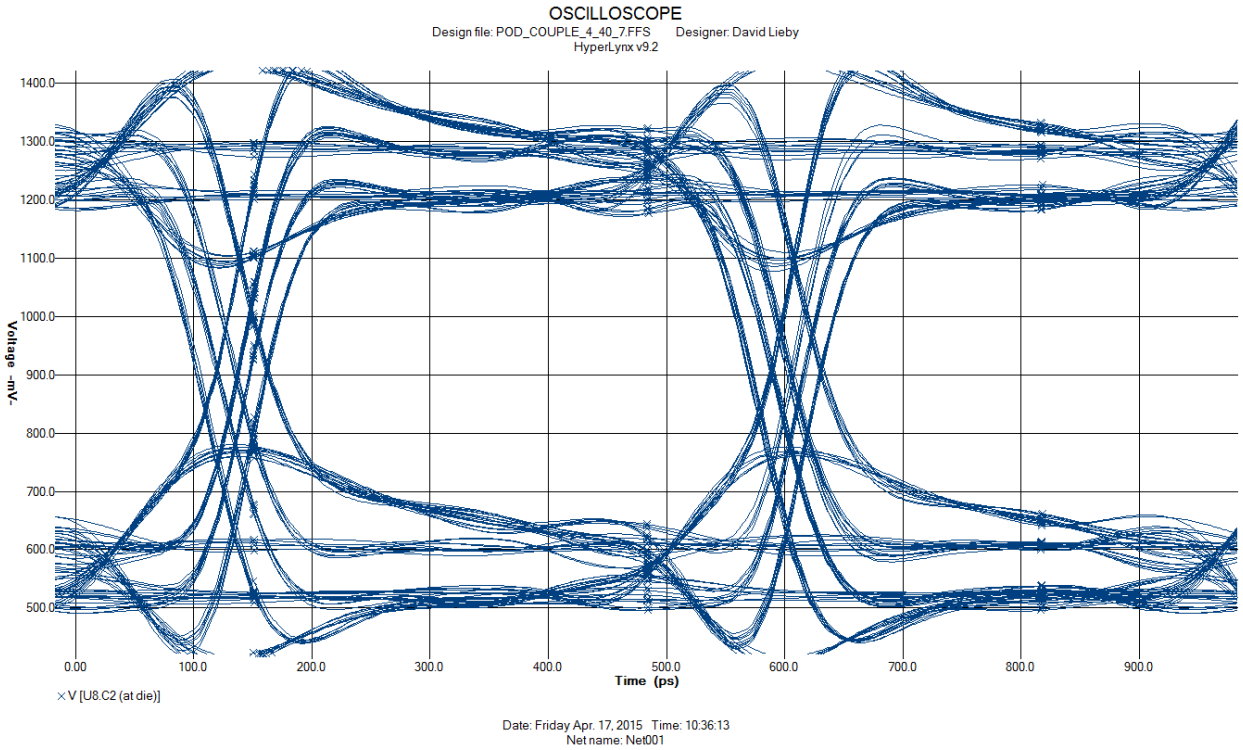
Channel Simulation for crosstalk effects

When we add the -20db spacing value determined from the crosstalk simulation above (7 mils) and run the simulations for ISI and ISI with crosstalk the signal looks fairly good. See the waveforms below.

The blue signal is ISI with aggressors quiet, the red signal is with aggressors running for a 3 inch topology. Aggressors for this are out of phase with the victim.



This plot is the same topology with the memory devices driving the circuit the difference here is the impedance of the FPGA package.



Channel Simulation for SSN effects

Simultaneous switching noise is defined here as the effects on the power supply of many signals switching together and the effect of this on the channel. This sort of simulation is not possible in HyperLynx LineSim.

Chapter 7: Further Investigations

Post Layout Simulation

In order to determine the overall usefulness of the layout a post-layout simulation of the channel should be executed.

It is important to use an IBIS model for the FPGA that is generated from a Quartus® II project that is created for the system. The IBIS model created this way will have correct values for the package parasitic values.

Further Reading

Altera has a wealth of information on developing and simulating DDR devices. The address and command signals for DDR4 are similar to those for DDR3 except for the voltage level of the interface.

EMIF Guidelines for Layout and Timing

http://www.altera.com/literature/hb/external-memory/emi_plan.pdf

Simulating for Timing Closure

The Altera wiki sight has a lot of information on this subject. Please refer to the following documents for detailed information on this subject.

The following links have a lot of information on timing parameters.

http://www.alterawiki.com/wiki/Arria_10_EMIF_Simulation_Guidance

http://www.alterawiki.com/wiki/Measuring_Channel_Signal_Integrity

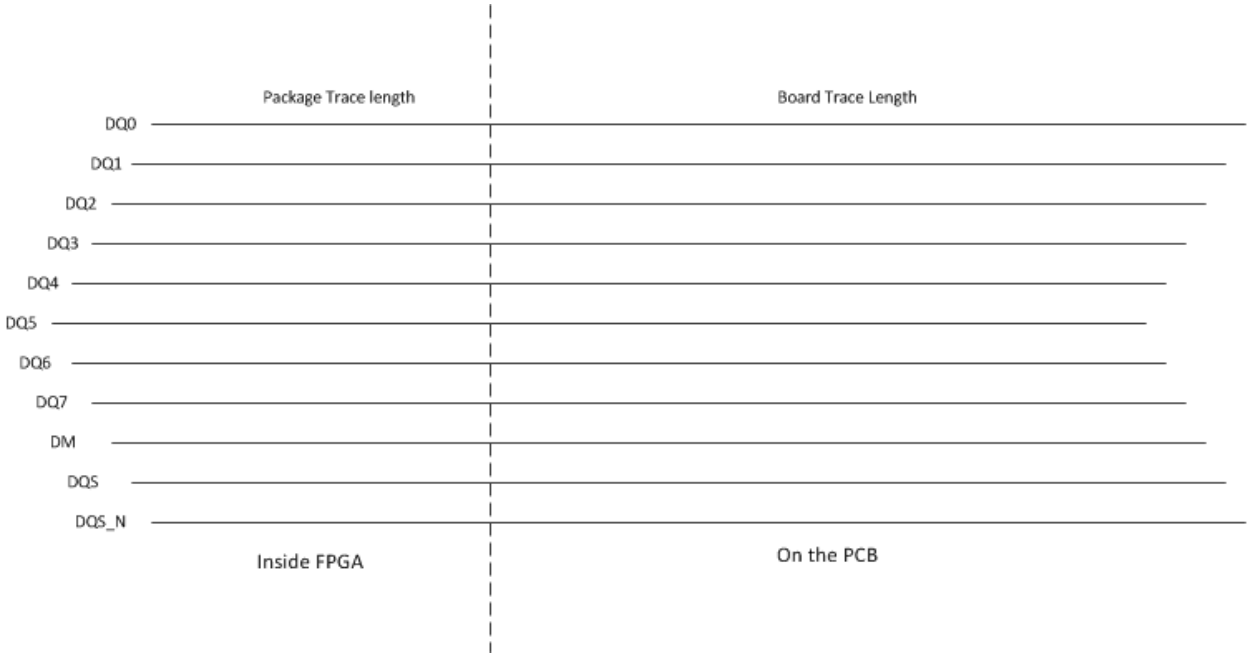
A definitive document on DDR4 voltages and timing is available from Micron Semiconductor. Understanding the POD training is very valuable to the designer. A full PDF of the datasheet has all the data in one place. Search on their website for “4Gb_DDR4_SDRAM” for a copy.

Check these things

Trace impedance plays an important role in the signal integrity. Users should perform board level simulation to determine the best characteristic impedance for their PCB. For example, it is possible that for multi rank systems 40 ohm would yield better result than the traditional 50 ohm characteristic impedance.

To minimize PCB layer propagation variance, Altera recommend that you route signals from the same net group on the same layer.

1. Use 45° angles (*not* 90° corners).
2. Disallow critical signals across split planes.
3. Route over appropriate VCC and GND planes.
4. Keep signal routing layers close to GND and power planes.
5. Avoid routing memory signals closer than 0.025 inch (0.635 mm) to memory clocks.
6. Match the (package + board) trace delays up to 20 ps of skew for DQ/DQS/DM signals within a DQS group.
7. Details on how to do package de-skew is available in EMIF HB vol2 chapter 4.



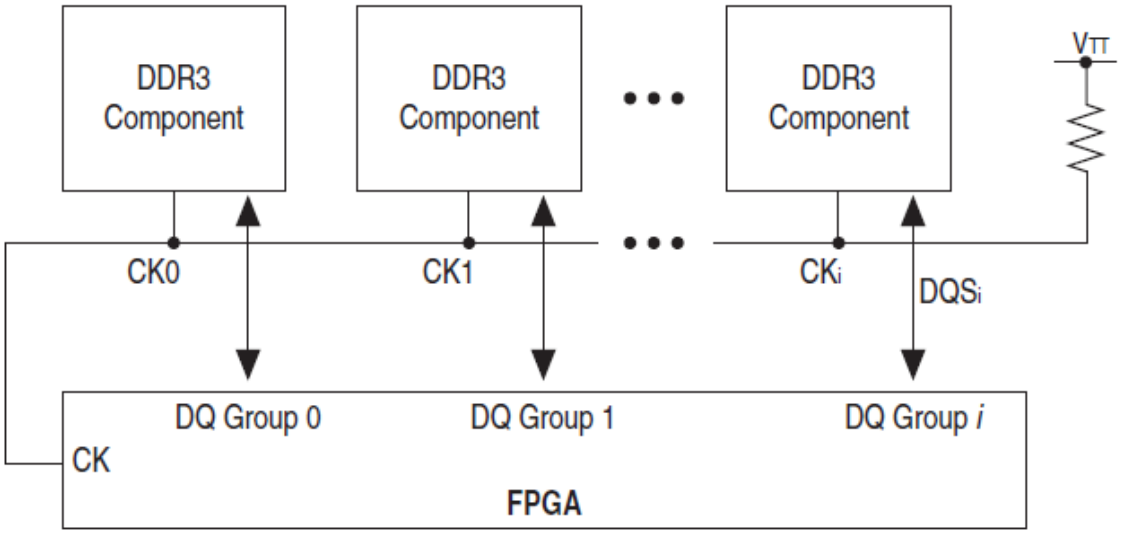
8. All the address, command and control signals should match up to +/- 20 ps compare to the mem_clk trace.
    - For example if the mem_clk trace delay is 500 ps then the allowed range for any address/command/control signal is 480 ps to 520 ps.

- For discrete components; make sure above recommendation is met for each component in the fly-by chain.
- For DIMMs: For single or multiple DIMM configurations make sure this guideline is met at each DIMM connector.

The timing between the DQS and clock signals on each device calibrates to meet tDQSS.

9. Make sure that DQS arrives after clock :
   - (CKi) – DQSi > 0; 0 < *i* < number of components – 1
10. Total skew of CLK and DQS signal between groups is less than one clock cycle:
   - (CKi + DQSi) max – (CKi + DQSi) min < 1 × tCK

If you are using a DIMM topology, your delay and skew must take into consideration values for the actual DIMM.



CK_i = Clock signal propagation delay to device *i*

DQS_i = DQ/DQS signals propagation delay to group *i*