SystemVision Example:

H-Bridge SPICE Motor Controller

H-Bridge SPICE Motor Controller

This help file discusses how to use SystemVision to simulate a simple H-Bridge Motor Controller. This example combines SPICE models for the electronics with VHDL-AMS models for the motor and loads. The ability to simulate these together is made possible by SystemVision's support for multiple industry standard languages.

The SystemVision project is named **Beyond_SPICE**. Contained in the project are several test benches that illustrate how SystemVision uses VHDL-AMS to extend SPICE beyond its inherent limitations for modeling mixed-technology systems. This help file will sequentially walk you through each of these designs.

Getting Started

Users are encouraged to look through the following documents from the SystemVision Help menu.

1. **Help > SystemVision Help Topics**
   This is the main online help resource with hyperlinks. Topics include: Introduction to SystemVision, The User Interface, SystemVision Commands, and "How do I …?" tips.
2. **Help > SystemVision Online Manuals > Getting Started**
   The Getting Started tutorial is provided to acquaint users with SystemVision. It is recommended that users run through this tutorial prior to running other examples.
3. **Help > SystemVision Online Manuals > User's Manual**
   The Users Manual is provided as a reference source for SystemVision. Detailed explanations on tool usage are provided.
4. **Help > SystemVision OnlineManuals > SystemVision Quick Reference Guide**
   This document should be printed and used as a reference for running SystemVision examples.
5. **Help > SystemVision Online Manuals > VHDL-AMS Quick Reference Guide**
   This VHDL-AMS reference succinctly reviews many VHDL-AMS language features. It is intended for those users interested in creating/understanding VHDL-AMS models.
6. For specific information on the Waveform Viewer, select **Help > Quick Start and Help > Tutorial > Open Tutorial** from the main Waveform Viewer window).

1.0 tb_hbridge_spice testbench

Description:

The design tb_hbridge_spice, shown in Figure 1, is an example of a typical hbridge driver circuit modeled in SPICE. Voltage pulse sources are used as the forward and reverse command signals and the motor load is approximated with a resister and inductor in series.

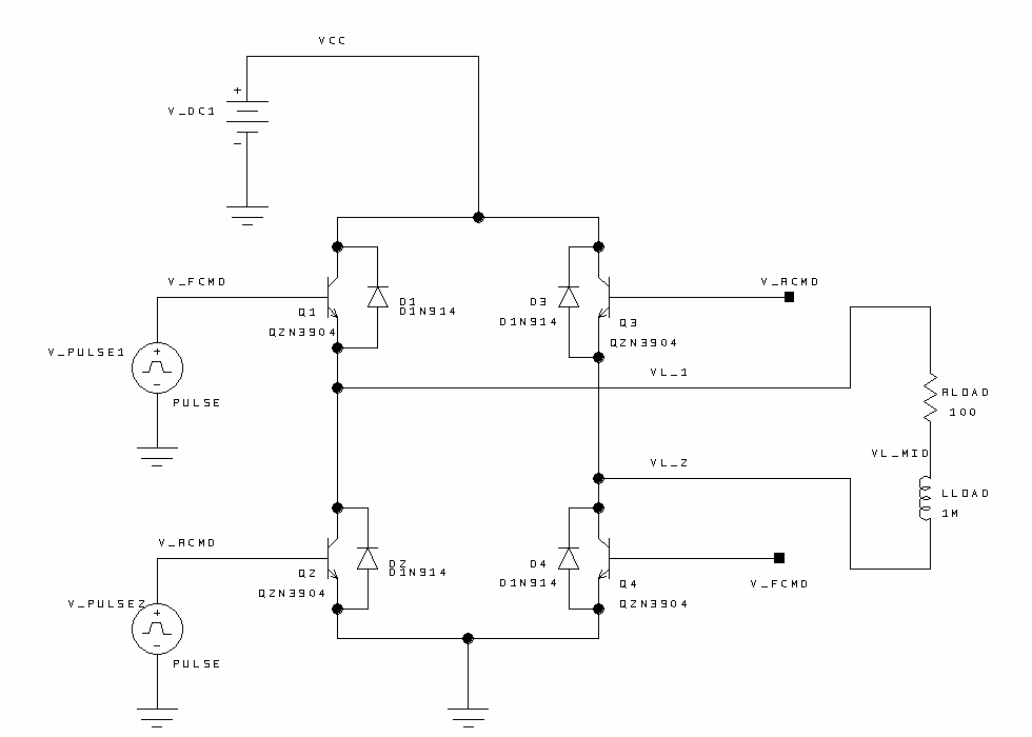


Figure 1- tb_hbridge_spice schematic

Using SPICE Primitives in SystemVision

Although SystemVision may be known primarily as a VHDL-AMS simulator, is it also capable of operating as a SPICE simulator. To do this, SystemVision can be configured to run in what is called "SPICE-on-Top" mode. Here the top-level netlist is SPICE and calls to models written in other languages (like VHDL-AMS) are done through an instantiation of a SPICE "Y" element. To facilitate this, schematic symbols have been created that reside in two symbol libraries:

1. **SpicePrimitive**: Symbols of basic SPICE passive elements (R,L,C), controlled sources (E,F,G,H) and others.
2. **SpiceSemiconductor**: Symbols for the 4 basic semiconductor devices (D, Q, M, J)

Using these symbols will automatically force a SPICE netlist to be created allowing the user to operate in a familiar mode even when adding VHDL-AMS models.

Design Selection/Viewing/Editing

1. Make sure the **Beyond_Spice** project is loaded into SystemVision.
2. From the **Project** tab, click on **Design Roots > tb_hbridge_spice** to open the schematic.

Accessing the Spice symbols

1. To access the SPICE symbols mentioned above, select **Add > Component** from the pull-down menu and scroll down the symbol library list as shown in Figure 2.

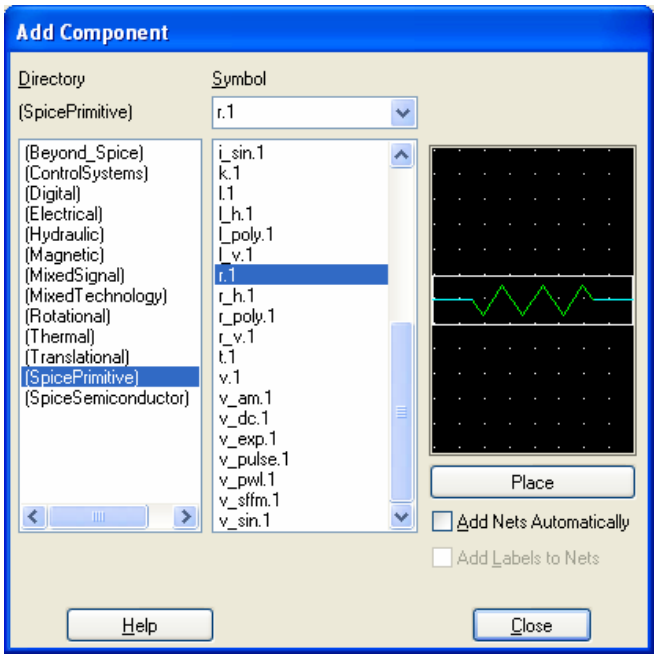


Figure 2 - Accessing SPICE Symbols

2. The schematic has already been created for you, so there is no need to place any symbols.

Simulating the circuit

3. From the Pull-down menu, select **Simulation > Simulate** to invoke the Simulation Control window (there is also a simulation icon available on the iconbar).
   - The **Simulations** tab is for the basic settings and to enable/disable the desired analyses. Enable the **Time-Domain Analysis** and set **End Time** to **1s**.
   - The **Results** tab allows you to select the signals for viewing. The default selection is **TopLevel Waveforms** which will save all signals at the highest level of hierarchy. Since this is a small design, **All Signals** was selected for convenience. You can also choose **Selected Waveforms** and click on the browse button (**...**) to select individual signals.
   - The **Multi-Run** tab is for performing parametric variations.
   - The **Advanced** tab is used for advanced simulation settings (normally not required).

4. Click the **OK** button to run the simulation (the **Output Window** will display the simulation progress and any error messages that may be issued).

Viewing Results

5. The simulation results will automatically be loaded into the waveform viewer upon completion of a successful simulation (if not, look in the **Output Window** for any error messages).
6. Individual signals can be viewed by double-clicking with the LMB or by using the drag-and-drop feature. You can also plot a signal from the schematic by selecting the desired signal and using **RMB > Plot Waveform**. Use one of these methods to plot the signals **v_fcmd, v_rcmd** and **vl_1** which should appear as shown in Figure 3 below:

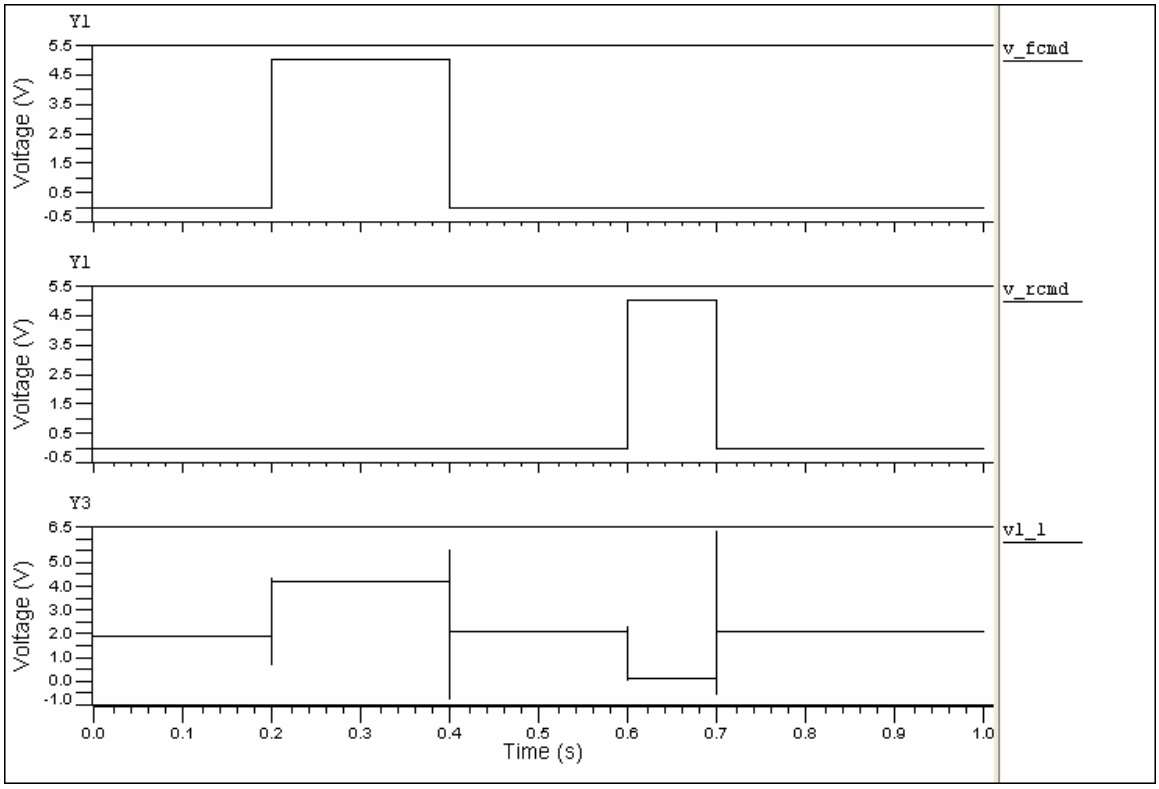


Figure 3 - tb_hbridge_spice signals

The forward and reverse command signals indicate the times the transistors are turned on, and the signal **vl_1** is the load voltage (with respect to ground). This is some good information, but what we really want is the current through the load. It is possible to view the currents through SPICE elements in SystemVision, but and extra step is required as explained in the next section.

Plotting SPICE currents in SystemVision

Plotting currents through SPICE elements in SystemVision is currently not supported when using the **Simulation Control** dialog. For this and other advanced commands, SystemVision allows you to edit the simulation command file. This file gives you direct access to the

simulation control language and provides an alternate method to running simulations that you will find useful.

1. Select the simulation tab in the **AddInTree** window on the left-hand side of the SystemVision interface.
2. Expand **tb_hbridge_spice** testbench to access the **Experiment Files** as in Figure 4 shown below:

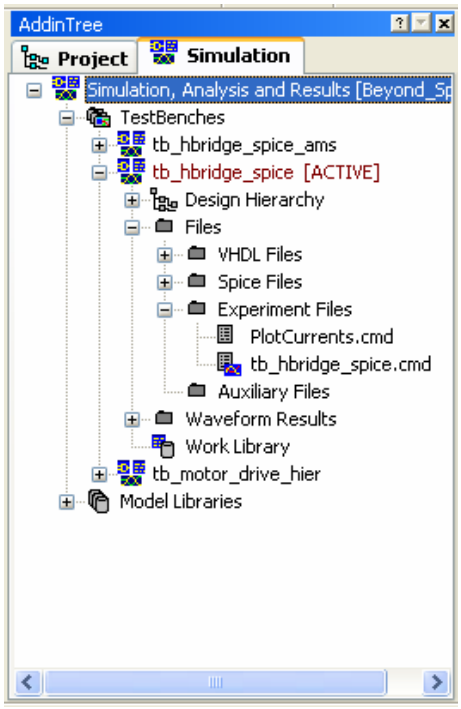


Figure 4 - Simulation Tab in SystemVision

3. Here an alternative experiment file has already been created for you (**PlotCurrents.cmd**).
4. Put your cursor over the **PlotCurrents.cmd** file and select **RMB > Open in Text Editor** to view.
5. Note the line **".probe v i "** gives you access to the voltages and currents through all elements.
6. The **File > Save As ...** command from the pull-down can be used to create multiple experiment files for a given design.
7. Put your cursor over the **PlotCurrents.cmd** file and select **RMB > Simulate** to run a simulation using this command file.
8. Upon completion, the simulation results will again be loaded into the viewer. Select the signal **i(rload)** and plot into the original graph window, replacing the **vl_1** signal as shown in Figure 5 below:

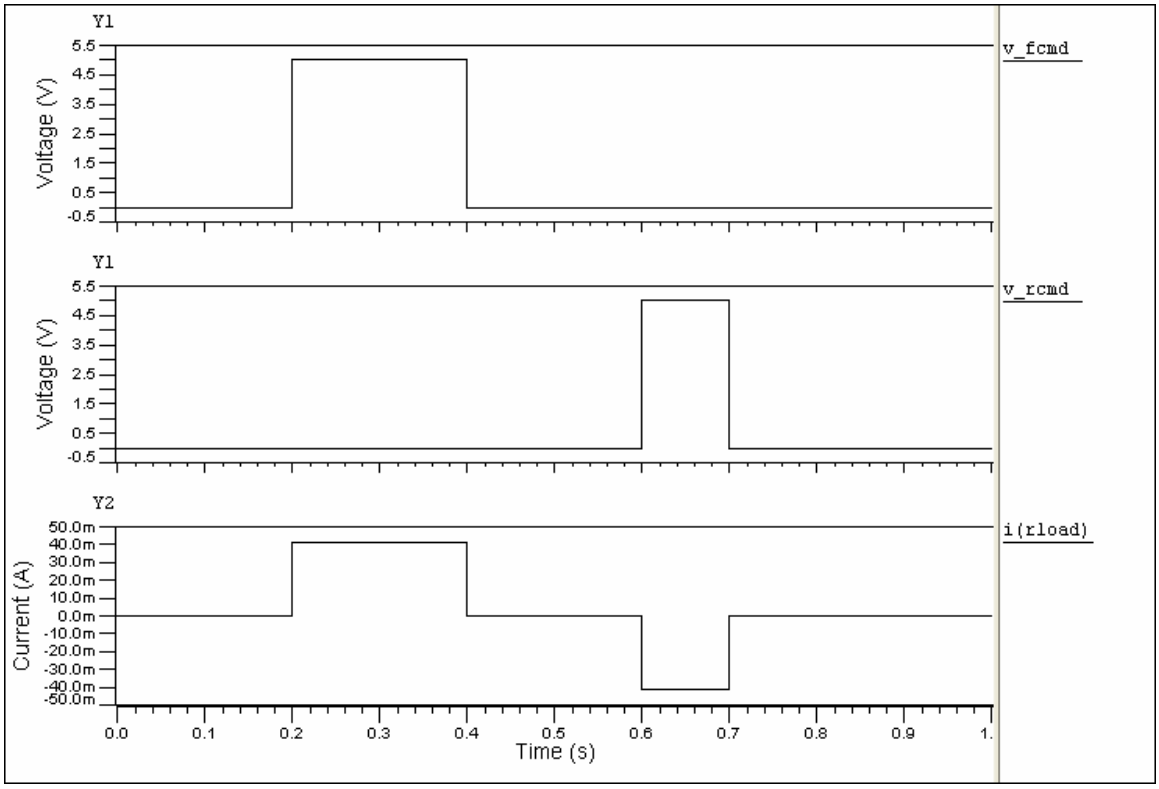


Figure 5 - Simulation results with load current

9. Plot any other signals of interest.

2.0 tb_hbridge_spice_ams testbench

Description

The design **tb_hbridge_spice_ams**, shown in Figure 6, is a modified version of the previous H-bridge circuit. The original transistor circuit implementation is still in tact, but the voltage command signals have been replaced by digital (VHDL) stimulus and the simplified RL load has been replaced by a VHDL-AMS DC motor model. The addition of these components makes it a much more realistic system model. Using SystemVision, the power of VHDL-AMS can be seamlessly incorporated to an existing SPICE circuit with no additional burden on the end user.

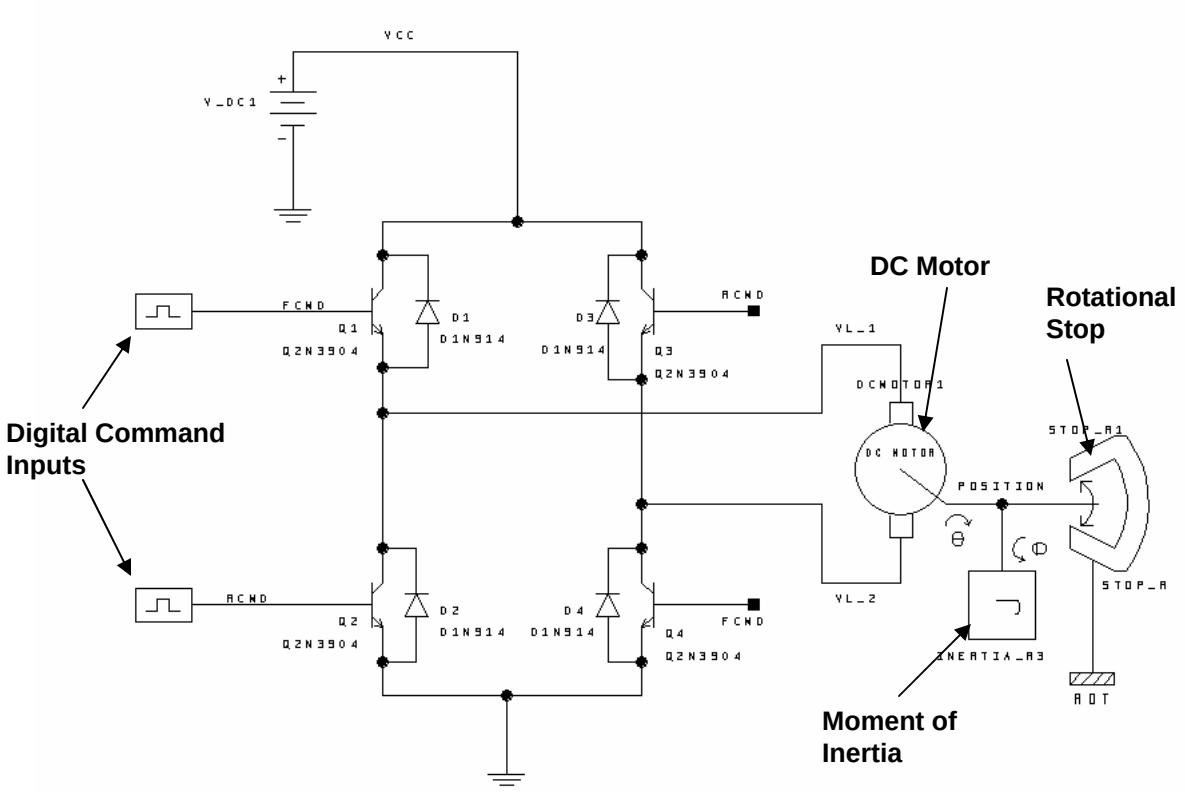


Figure 6 - tb_hbridge_spice_ams schematic

Design Selection/Viewing/Editing

1. Make sure the **Beyond_Spice** project is loaded into SystemVision.
2. From the **Project** tab, click on **Design Roots > tb_hbridge_spice_ams** to open the schematic.
3. Select the DC Motor Symbol and **RMB > Push Language > VHDL** to view the VHDL-AMS model (repeat for other VHDL-AMS models as desired)
4. **RMB > Edit Model Properties** gives you access to the VHDL-AMS model properties.

Simulating the circuit

5. Return to the schematic and from the Pull-down menu, select **Simulation > Simulate** to invoke the Simulation Control window (there is also a simulation icon available on the iconbar).
6. The **Simulations** tab is for the basic settings and to enable/disable the desired analyses. Enable the **Time-Domain Analysis** and set **End Time** to **1s**.
   - The **Results** tab allows you to select the signals for viewing. The default selection is **TopLevel Waveforms** which will save all signals at the highest level of hierarchy. Since this is a small design, **All Signals** was selected for convenience. You can also choose **Selected Waveforms** and click on the browse button (**...**) to select individual signals.
   - The **Multi-Run** tab is for performing parametric variations.
   - The **Advanced** tab is used for advanced simulation settings (normally not required).

7. Click the **OK** button to run the simulation (the **Output Window** will display the simulation progress and any error messages that may be issued).

Viewing Results

8. The simulation results will automatically be loaded into the waveform viewer upon completion of a successful simulation (if not, look in the **Output Window** for any error messages). Individual signals can be viewed by double-clicking with the LMB or by using the drag-and-drop feature. You can also plot a signal from the schematic by selecting the desired signal and using **RMB > Plot Waveform**. Use one of these methods to plot the digital command signals **fcmd** and **rcmd** indicated by the ⎍ symbol next to the waveform name.
9. Next plot the motor current **ydcmotor1/i**. Notice the effect of the back-emf voltage as modeled in the motor equations.
10. Plot the signal **position**. This is the position of the motor shaft over time. Notice that the motor does not return to its original position due to the short length of the reverse command signal.
11. The desired waveforms are shown in Figure 7 below:

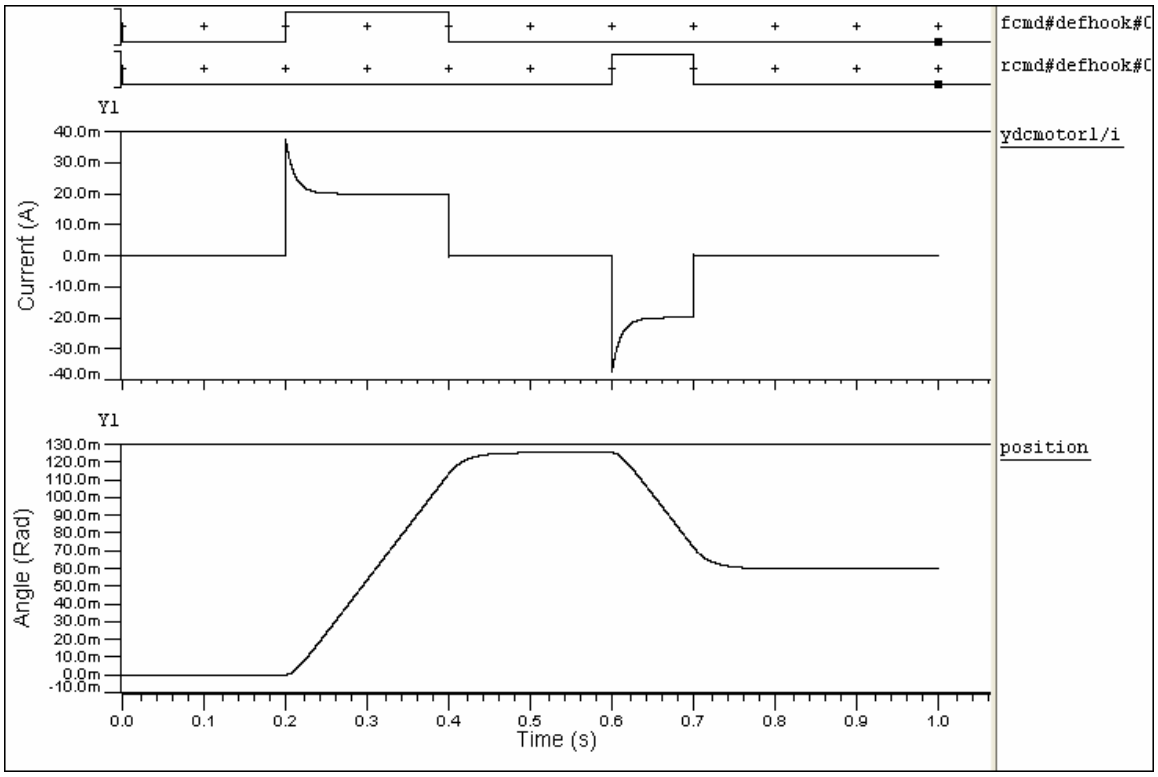


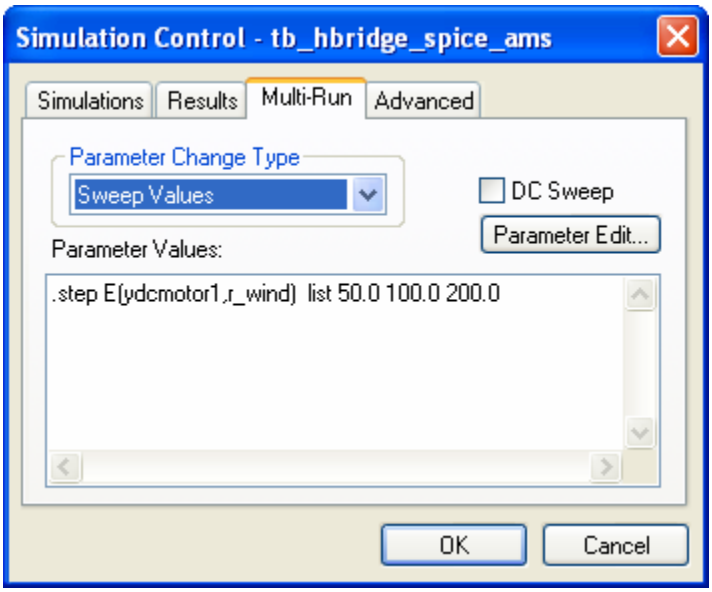
Figure 7 - Motor current and position

Parametric Sweeps

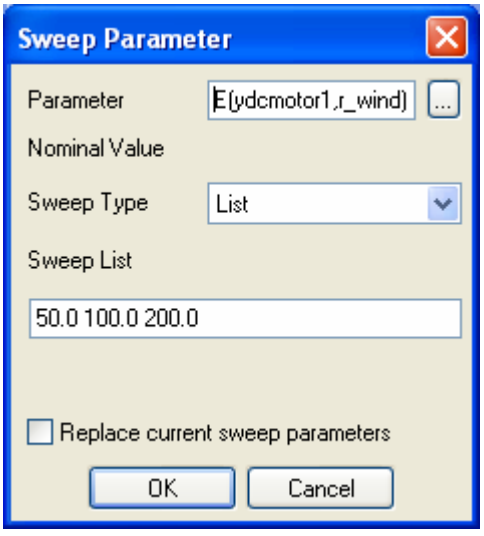
This example illustrates that by adding VHDL-AMS models to a SPICE circuit, we are able to easily create a mixed-technology system model. Using this model we are not limited to the electrical domain. For example, we are able to view the motor shaft angular position and looks at the effect of the back-emf voltage generated by the motor.

The next step is to look at this system as a whole and examine the effects of the component parameters on the motor position. We will do this by sweeping a parameter (motor winding resistance) over a specified range.

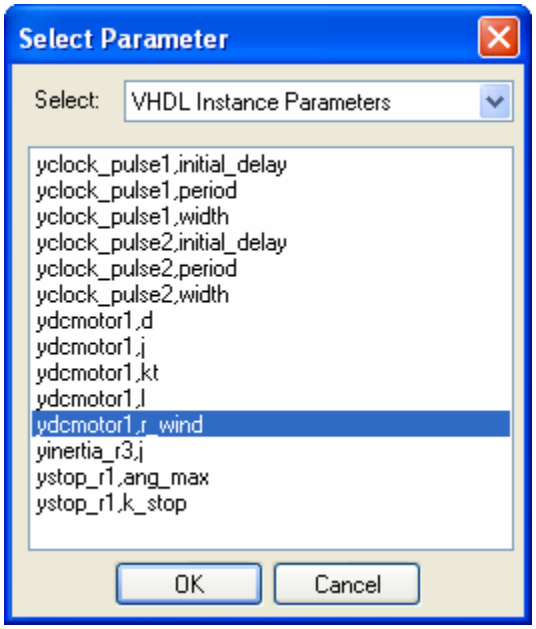
1. Return to the **tb_hbridge_spice_ams** schematic and from the pull-down menu, select **Simulation > Simulate** to invoke the Simulation Control window
2. Select the **Multi-Run** tab as shown below:



3. Change the **Parameter Change Type** to Sweep Values
4. Click on the **Parameter Edit** button to invoke the **Sweep Parameter** dialog:



5. Click on the browse button (**...**) to invoke the **Select Parameter** dialog:



6. Select the **VHDL Instance Parameter** ydcmotor1.r_wind as shown above and click **OK**.
7. Set the **Sweep Type** to **List** and the **Sweep List** to 50.0 100.0 200.0.
8. Click **OK** to save settings and run the simulation.
9. Plot the ***digital*** command signals (**fcmd** and **rcmd**) along with the motor position as shown in Figure 8 below:

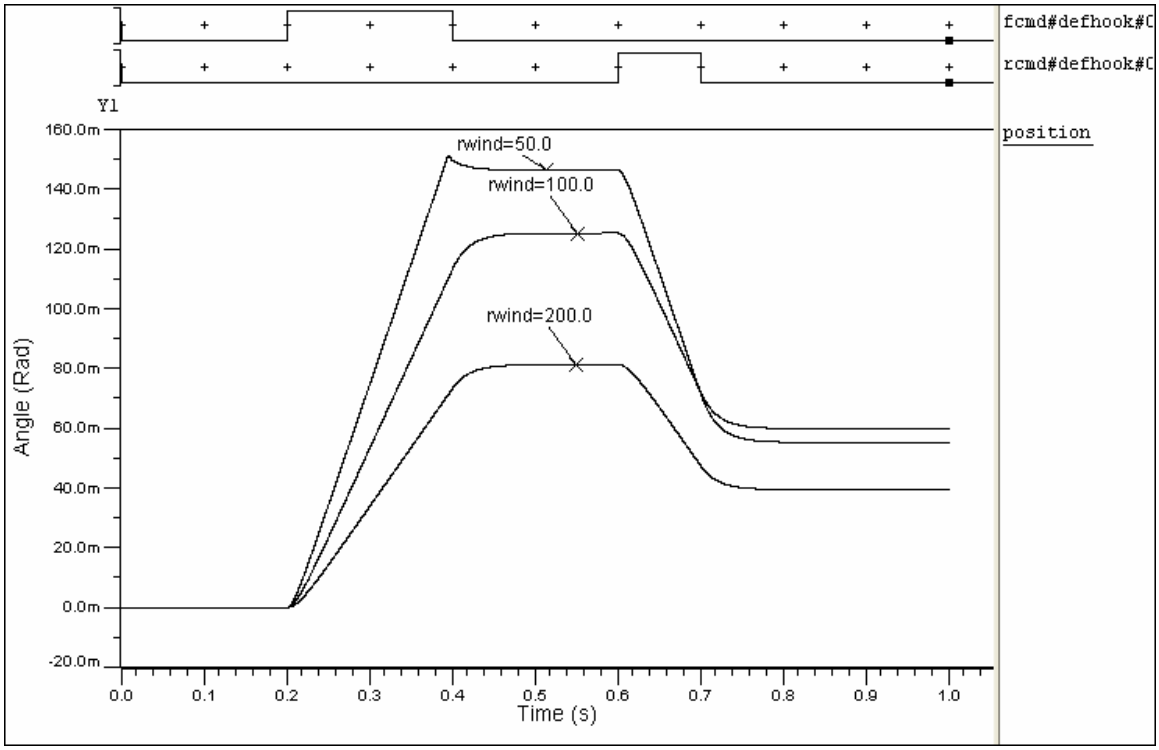


Figure 8 - Parametric Sweep Results

Performing Measurements (optional)

As expected, these results show that the winding resistance has a significant effect on the final angular position. SystemVision allows you to quantify relationships between system performance and component parameters by performing measurements on the simulation results. In this section we will re-run the previous analysis with additional sweep values. This will take a bit longer to run, but the extra data generated will allow us to create meaningful correlation plots.

1. Re-invoke the **Simulation Control** dialog and click on the Multi-Run tab
2. Click the **Parameter Edit** button and modify the existing fields as follows:
   Parameter: E(ydcmotor1,r_wind) (unchanged from previous analysis)
   Sweep Type: Increment
   Sweep Start: 10.0
   Sweep Stop: 100.0
   Sweep Increment: 10.0
3. Plot the signal **position** into a new graph
4. Click the **Add Cursor** button to place two cursors (**C1** and **C2**) as shown in Figure 9 below:

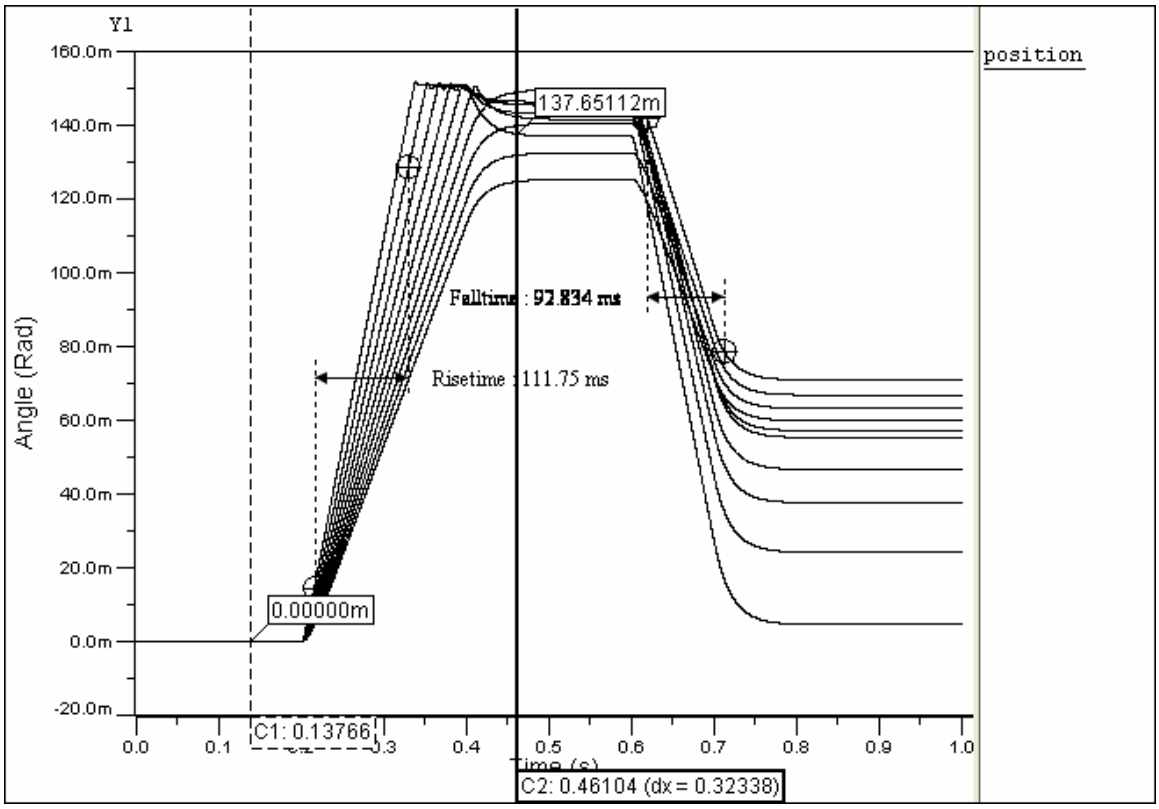
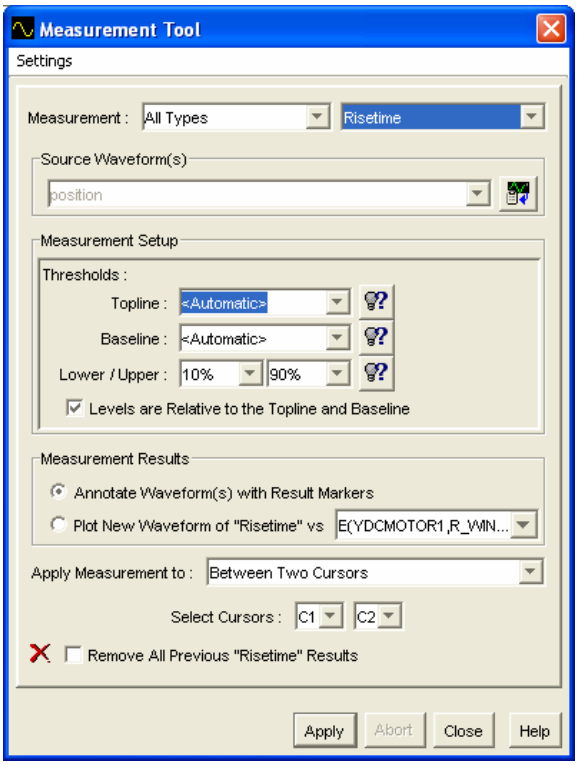


Figure 9 - Parametric Sweep Results with Measurements

5. Click the **Measurement Tool** icon and fill in the dialog as shown below to annotate the **risetime** measurement results to the graph:



6. In the **Measurement Tool**, select the **Plot New Waveform ...** option and re-apply to create a plot of risetime vs. motor winding resistance.
7. Repeat steps 5 and 6 above for **falltime**.
8. The results are shown in Figure 10 below:

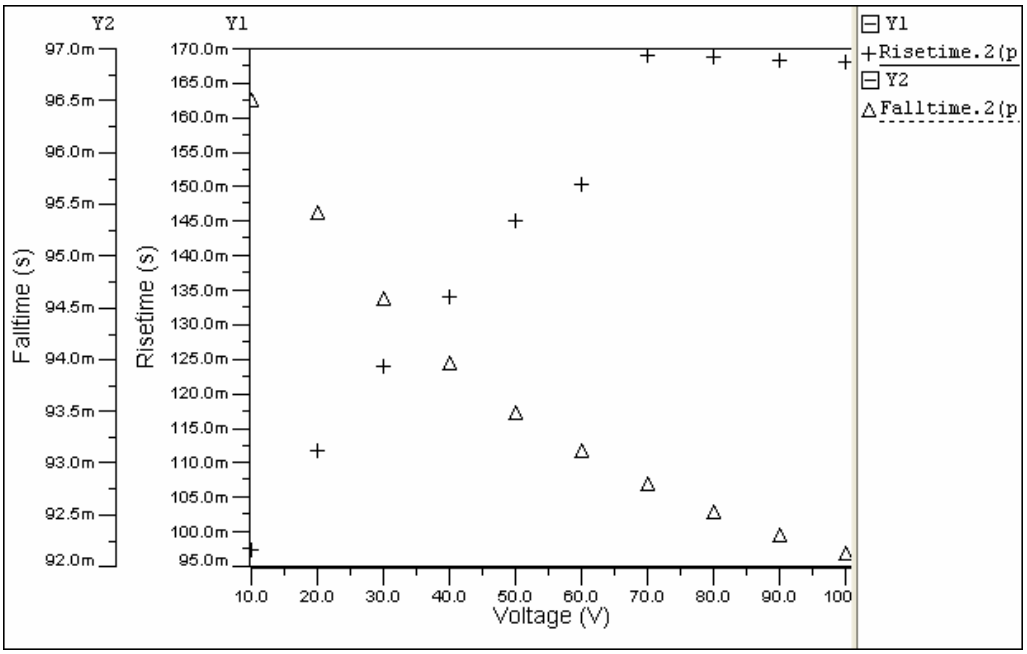


Figure 10 - Correlation Plots

Figure 10 clearly shows the effect the value the motor winding resistance has on the shaft position when used with the H-bridge driver circuit. This is a simple example of how SystemVision can be used to turn simulation data into engineering information that can be used to help improve a mixed-technology system design.

3.0 tb_motor_drive_hier testbench

Description

The design **tb_motor_drive_hier** shown in Figure 11 is identical to the previous design except that the H-bridge circuit has been imported as a SPICE netlist using the symbol HBRIDGE1. Using SystemVision any Berkeley-compatible SPICE netlist can be imported into SystemVision and directly connected to VHDL-AMS component models.

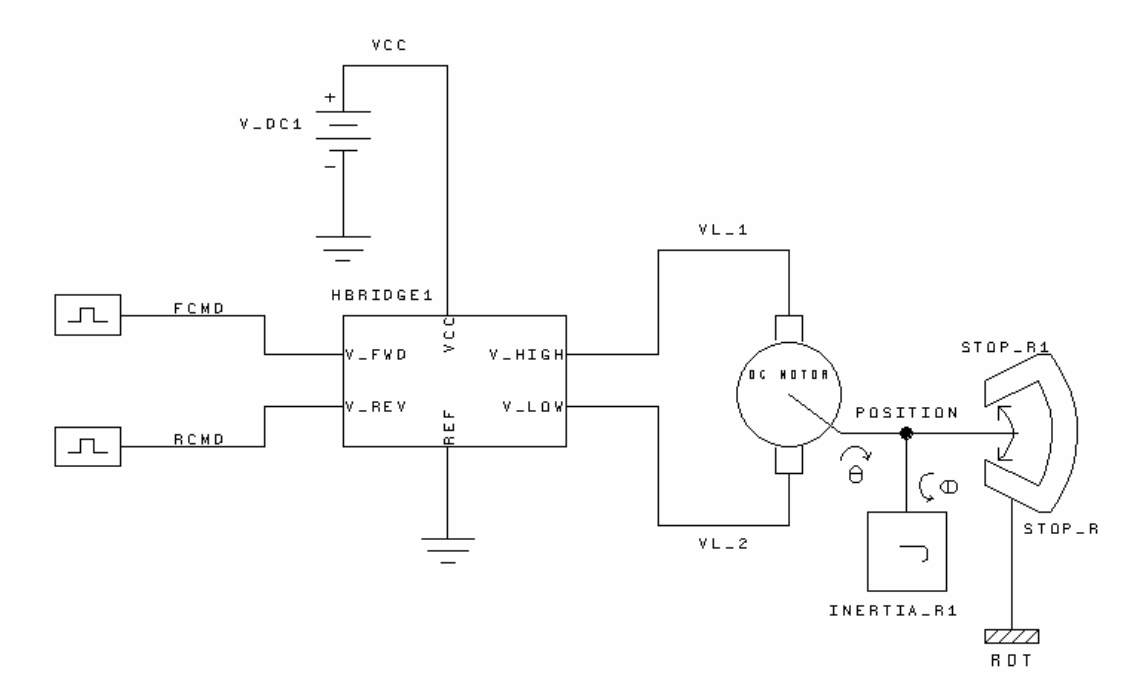


Figure 11 - Motor Drive System with SPICE netlist for H-Bridge Circuit

Design Selection/Viewing/Editing

1. Make sure the **Beyond_Spice** project is loaded into SystemVision.
2. From the **Project** tab, click on **Design Roots > tb_motor_drive_hier** to open the schematic
3. Select the **HBRIDGE1** symbol and **RMB > Push Language->SPICE** to reveal the SPICE netlist shown in Figure 12 below:


```
*  Definition for project MOTOR_DRIVE
.SUBCKT MOTOR_DRIVE REF VCC V_FWD V_HIGH V_LOW V_REV
*
Q1 VCC V_FWD V_HIGH Q2N3904
Q2 V_HIGH V_REV REF Q2N3904
Q3 VCC V_REV V_LOW Q2N3904
Q4 V_LOW V_FWD REF Q2N3904
D1 V_HIGH VCC D1N914
D2 REF V_HIGH D1N914
D3 V_LOW VCC D1N914
D4 REF V_LOW D1N914
* CROSS-REFERENCE 1
* GND = 0

.model D1N914 D(Is=0.1p Rs=16 CJO=2p Tt=12n Bv=100 Ibv=0.1p)
*          85-??-??   Original library
.model Q2N3904 NPN(Is=6.734f Xti=3 Eg=1.11 Vaf=74.03 Bf=416.4 Ne=1.259
+ Ise=6.734f Ikf=66.78m Xtb=1.5 Br=.7371 Nc=2 Isc=0 Ikr=0 Rc=1 Cjc=3.638p
+ Mjc=.3085 Vjc=.75 Fc=.5 Cje=4.493p Mje=.2593 Vje=.75 Tr=239.5n Tf=301.2p
+ Itf=.4 Vtf=4 Xtf=2 Rb=10)
*               Fairchild        pid=23          case=TO92
*               88-09-08 bam    creation
.ENDS
```

Figure 12 - H-Bridge SPICE Netlist

Simulating the circuit

Repeat the simulation steps in Section 2.0 above to verify that the functionality is identical.

This concludes the tutorial. For more information on using SPICE within SystemVision consult the following references accessible from the Help pulldown menu in SystemVision:

1. **Help > SystemVision Help Topics > How Do I …? > Files and Projects**
    a. **Import a SPICE subcircuit**
2. **Help > SystemVision Help Topics > How Do I …? > Models, Libraries, and Schematics**
    a. **Add and Use a SPICE Library**
    b. **Place a SPICE component or subcircuit in my design**
    c. **Create a symbol for SPICE subcircuit file**
    d. **Use a FAS (SPICE) macromodel**
3. **Help > SystemVision Help Topics > How Do I …? > Simulation**
    a. **Simulate a single SPICE component or design**
4. **Help > SystemVision Help Topics > Reference Information > SPICE Usage**
5. **Help > SystemVision On-Line Manuals > User's Manual**
    a. **Appendix A – Using Spice with SystemVision**

