SPICE Netlist Simulation Using NI LabVIEW and NI Multisim

Estimated time to complete: 45 minutes

Introduction

This tutorial demonstrates the step-by-step process to run text-based SPICE netlist simulations from **NI LabVIEW**. **NI Multisim** is used as the simulation engine and it is run in a background process connected to LabVIEW using the **LabVIEW Multisim API Toolkit**.

The LabVIEW Multisim API Toolkit is installed by Multisim and it provides over 80 VIs to control a Multisim application instance from LabVIEW. In this tutorial, you will explore the functionality of the SPICE command line interface this toolkit offers.

Tip Consult the toolkit Help to find programming guides, flowcharts and the toolkit VI reference.

- From LabVIEW, select **Help»NI Multisim API Toolkit Help**.
- From Multisim, select **Help»Multisim help**, and select the **Automation API** topic.

Objectives

- Create and modify a netlist file from a Multisim circuit file
- Create and modify a SPICE nutmeg command file
- Use LabVIEW to build a VI that will execute the SPICE nutmeg command file
- Display results from the simulation

There are multiple use-cases for this type of simulation, especially tailored for advanced SPICE users. Some of them may range from old library of SPICE files that you wish to simulate, make a batch process to run analyses on a directory of files, dynamically modify netlist files or analysis parameters to see differences, and so on.

This demo script assumes that the user has working knowledge of LabVIEW and Multisim, and understands the basics of SPICE. Even though the LabVIEW code in this tutorial looks simple, it is in fact intended for advanced users that are comfortable working with SPICE netlist and analyses.

Requirements

- NI LabVIEW 2013 or higher
- NI Multisim 14.0 or higher (Power Pro or Education Editions)
- LabVIEW Multisim API Toolkit (installed by Multisim)
- Files in the `SPICE Command Line` folder located in the Multisim `samples` folder

Note In a typical Multisim installation, the `samples` folder is in this location:

`C:\Users\Public\Documents\National Instruments\Circuit Design Suite <version>\samples\LabVIEW Multisim API Toolkit\`

Another way to find the `samples` folder in Multisim is to select **File»Open samples**, and take note of the folder location in the explorer window.

Create your own vs. use sample files

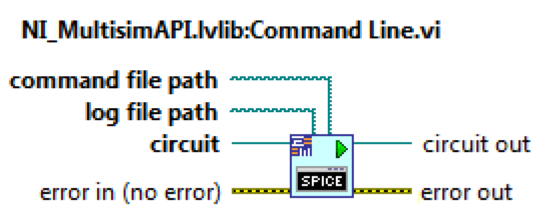
If you would like to follow this tutorial and create your own files all you need to start is the `SPICE Command Line.ms*` Multisim design file from the `samples` folder. Then follow the implementation instructions of this tutorial.

If you just want to quickly see the demonstration and/or study the sample code provided, grab all the files in the `SPICE Command Line` folder, and run the `SPICE Command Line.vi` LabVIEW VI.

Tip We recommend that you try to create your own files to get the most out of this exercise.

Command Line VI

In the implementation section later in this tutorial you will review the necessary code to connect to Multisim, simulate and then disconnect. However, the particular VI that will run the SPICE analysis is called **Command Line** VI.



This VI is found in the functions palette, under the **Connectivity » Multisim » Simulation and Analyses** subpalette. The short name displayed in the subpalette for this VI is **Command Line**.

Two file paths are needed as terminals. These are the **Command File** and the **Log File**. You will create and/or specify these paths later.

- **Command File—**this is a SPICE nutmeg command file, which includes instructions on what SPICE netlist to simulate, what analysis to run (and its parameters), and where to store the results. You will create this file with information from Multisim.
- **Log File**—this is a text file created during simulation, it lists every single command sent to the SPICE engine and the outcome of the command.

Two other files are involved in a SPICE command line simulation, the SPICE **netlist** and the **raw data** files. These two file paths are specified inside the Command File.

- **SPICE Netlist File—**this is the circuit description as a regular text-based SPICE netlist. You will create this file from Multisim.
- **Raw Data File**—this is a text file created during simulation, which stores the simulation results. You will parse this file to get the results.

The Command Line VI does not return the simulation results in a convenient LabVIEW-formatted data structure like other API toolkit VIs (for example, Waveform format). Inside the Command File you will specify the file path of the Raw Data file where the results will be saved in text format. You will then use LabVIEW File I/O and String VIs to properly parse the Raw Data file.

Standard SPICE netlist files from sources other than Multisim can also be used. Typically, you will see in the same netlist the circuit description and the analysis to run. To use the Command Line VI for that type of file, we suggest that you split such netlist into two files, one that only describes the circuit (SPICE Netlist File), and a second file that describes the analysis to run (Command File).

Implementation

This section contains the step-by-step instructions to create your own SPICE netlist files and the VI.

Numbered lines provide the high-level task. The square-bulleted lines explain in greater detail the steps needed to accomplish that task.

Multisim

1. Get all the files.

   - ❑ Group all the files included in the folder of this demo script into a new directory. In this way you will be able to modify them without compromising the originals.

   - ❑ If you are going to create your own files, only use the `SPICE Command Line.ms*` Multisim design file. The tutorial instructions are designed for this case.

2. Load the circuit file.

- Launch **Multisim**.
- Select **File » Open**, and select the `SPICE Command Line.ms*` file.
- Click **Open** to load the file and review the circuit.

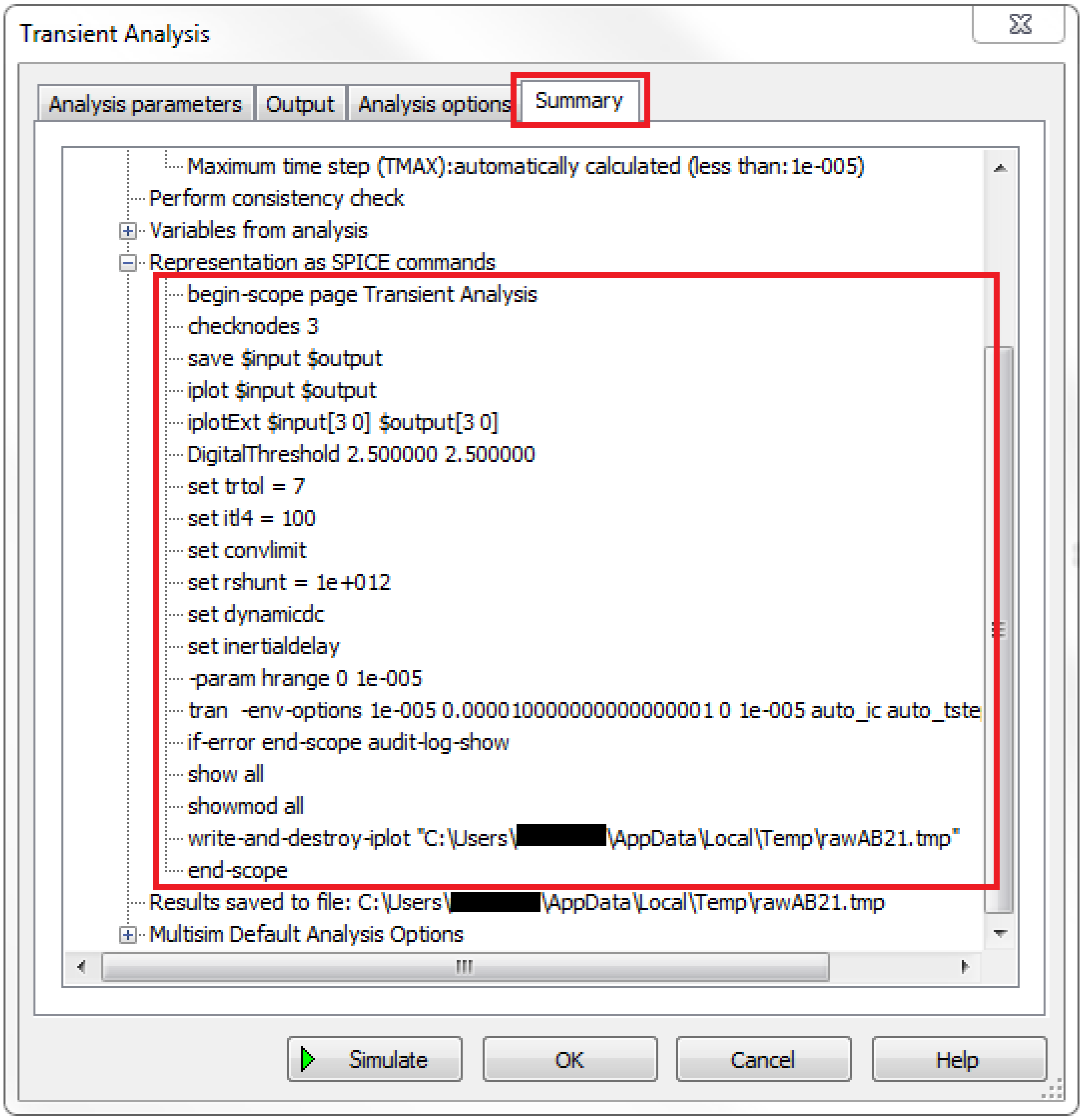
3. Create the SPICE netlist file.

   - Select **Transfer » Export SPICE netlist**.
   - Leave the default name or use `SPICENetlist.cir`. Navigate to the same folder as the circuit file.
   - Click **Save**.
   - You can open the file in Notepad® if you wish to review the results of this export; it is a text file. This is a typical SPICE netlist created by Multisim from the loaded circuit.

4. Create a sample SPICE nutmeg command file.

   - Select **Simulate » Analyses » Transient analysis**.
   - Select the **Output** tab, and confirm that `V(output)` and `V(input)` are listed inside **Selected variables for analysis**.
   - Select the **Summary** tab, and expand the **Representation as SPICE commands** branch.
   - Launch Notepad® or any other text editor, and type all the command lines starting from `begin-scope` to `end-scope`.
   - You should end up with text similar to the following code snippet:

```
begin-scope page Transient Analysis
checknodes 3
save $input $output
iplot $input $output
iplotExt $input[3 0] $output[3 0]
DigitalThreshold 2.500000 2.500000
set trtol = 7
set itl4 = 100
set convlimit
set rshunt = 1e+012
set dynamicdc
set inertialdelay
-param hrange 0 1e-005
tran -env-options 1e-005 0.000010000000000000001 0 1e-005 auto_ic
auto_tstep auto_tmax
if-error end-scope audit-log-show
show all
showmod all
write-and-destroy-iplot "RawData.tmp"
end-scope
```



- ❑ In Notepad®, select **File** » **Save**. Enter the name `SPICENutmeg.cir` for the file and browse to the same directory as the netlist file of **Step 3**.
- ❑ Click **Save**.
- ❑ Close Multisim.

Note This tutorial shows a transient analysis. If you need to run any other SPICE analysis, in particular those that are not included in the LabVIEW Multisim API Toolkit, you can get the correct SPICE nutmeg commands from the same **Summary** tab on any Multisim analysis.

5. Modify the SPICE nutmeg command file.

   There are multiple sections that need to be modified from the nutmeg command file before it can be used.

   - ❑ Specify the source SPICE netlist file. Enter the following code *before* the `begin-scope` line shown below:

```
source "C:\MyDirectory\SPICENetlist.cir"
```

Where `MyDirectory` is the location in your disk where the exported SPICE netlist file from **Step 3** was saved to.

One of the subVIs included in the tutorial folder can create local paths. If you don't want to hard-code the file path, you can let the subVI (explained later) take care of the paths. If you are using this subVI then just leave the filename with extension inside the quotes (no full path), and also make sure that the SPICE netlist file was saved as `SPICENetlist.cir` for this subVI to work.

Note If you are hard-coding file paths, use the full file path to the SPICE netlist file and enclose it with double quotes (“”).

   - ❑ Notice in the SPICE nutmeg command file, the selected variables in Multisim to simulate were renamed from `V(output)` and `V(input)` to `$output` and `$input`, this is an internal convention used in the Multisim engine. You will see these two variables listed in the `save` and the `iplot` lines in the code.

   - ❑ Specify the raw data results file so that it is located within the same directory as your `*.cir` files:

```
write-and-destroy-iplot "C:\MyDirectory\RawData.tmp"
```

If you are going to use the tutorial subVI for creating local paths then just leave `RawData.tmp`.

   - ❑ The final nutmeg file should look like the following code snippet.

```
source "C:\MyDirectory\SPICENetlist.cir"
begin-scope page Transient Analysis
checknodes 3
save $input $output
iplot $input $output
iplotExt $input[3 0] $output[3 0]
DigitalThreshold 2.500000 2.500000
set trtol = 7
set itl4 = 100
set convlimit
set rshunt = 1e+012
set dynamicdc
set inertialdelay
-param hrange 0 1e-005
tran -env-options 1e-005 0.000010000000000000001 0 1e-005 auto_ic
auto_tstep auto_tmax
if-error end-scope audit-log-show
show all
showmod all
write-and-destroy-iplot "C:\MyDirectory\RawData.tmp"
end-scope
```

The file paths should point to either a hard-coded path, or just to the filenames indicated if you are planning to use the tutorial subVI.

This SPICE nutmeg command file will execute the analysis commands, and is the **Command File** used by the **Do Command Line** VI. The `source` line specifies which circuit netlist file to load and to apply the analysis to. The analysis results are saved to the file specified in the `write-and-destroy-iplot` line.

- ❑ Save the changes and close Notepad®.

LabVIEW

6. Start LabVIEW.

   - ❑ Launch **LabVIEW**.
   - ❑ Confirm that you have the **LabVIEW Multisim API Toolkit VIs**, they should be located under the **Connectivity » Multisim** subpalette.

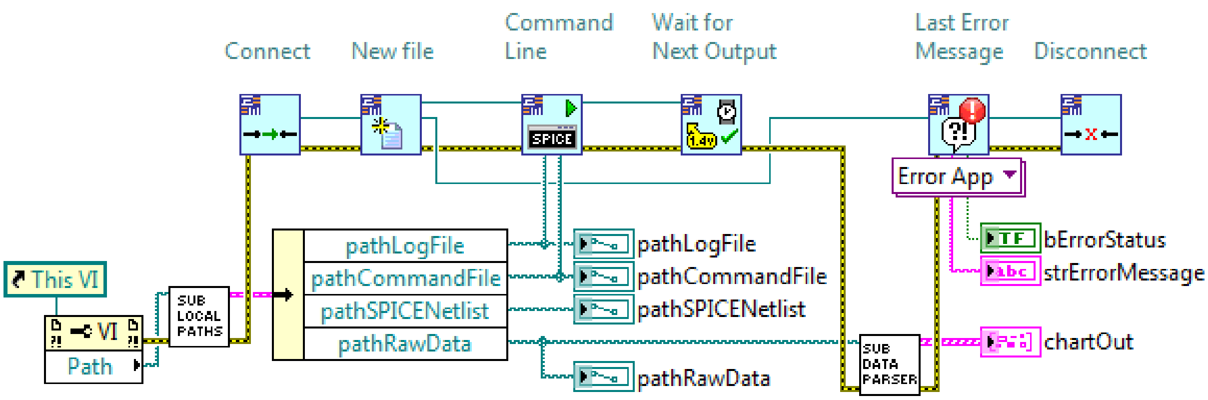
Note If you do not see the toolkit VIs, you may need to install the LabVIEW Multisim API Toolkit. It can be installed from the Multisim installer. Select the toolkit from the features to install tree. LabVIEW 2013 or higher needs to be installed for the Multisim installer to enable the toolkit feature.

7. Build the VI.

- ❑ Select **Help » NI Multisim API Toolkit Help**.
- ❑ In the Help file, browse to **Programming Guide » Programming Flowcharts » Flowchart for SPICE Netlist**.
- ❑ Use the flowchart as a reference to build the LabVIEW VI that will execute the SPICE netlist.

Tip If you would rather use a pre-built VI, open and run the `SPICE Command Line.vi` included with this tutorial. It has more code than the example shown here but the logic is similar.

This is the **minimum code** you need to build:



The LabVIEW Multisim API Toolkit VIs are located in the functions palette. Right-click the block diagram and select the **Connectivity » Multisim** subpalette. Use the following table if you are having trouble finding the VIs in the subpalettes.

VI	Subpalette
Connect	Connection
New File	File
Command Line	Simulation
Wait for Next Output	Simulation
Last Error Message	Error & Utility
Disconnect	Connection

Two subVIs are included with this tutorial, **sub Create Local Paths** VI and **sub Data Parser** VI, which are custom made, you can create your own if you wish, or for convenience use these subVIs provided.

- **sub Create Local Paths**—this VI takes the caller VI local path and creates four file paths in the same directory, one path each for the log file, command file, SPICE netlist file and the raw data results file.

- **sub Raw Data Parser**—this VI finds the raw data file and parses it into a LabVIEW XY Graph display. It is custom made for this particular transient analysis with two variables.

Tip Review the sub Raw Data Parser VI in particular, as it provides a reference on how to properly parse the raw data results file.

Since you have not run the VI yet, you do not have the raw data results file to reference. You can use the `RawData.tmp` file included with this tutorial to see how Multisim formats the results in the text file. You may open this file in Notepad® or any text editor program. The sub Raw Data Parser VI will be easier to understand.

The **Wait for Next Output** VI can be replaced by more preferred architectures, like event-based architectures. It is the easiest to include in a small program like this example VI.

Error-checking is not required, but is highly recommended. The **Last Error Message** VI helps to troubleshoot and manage errors.

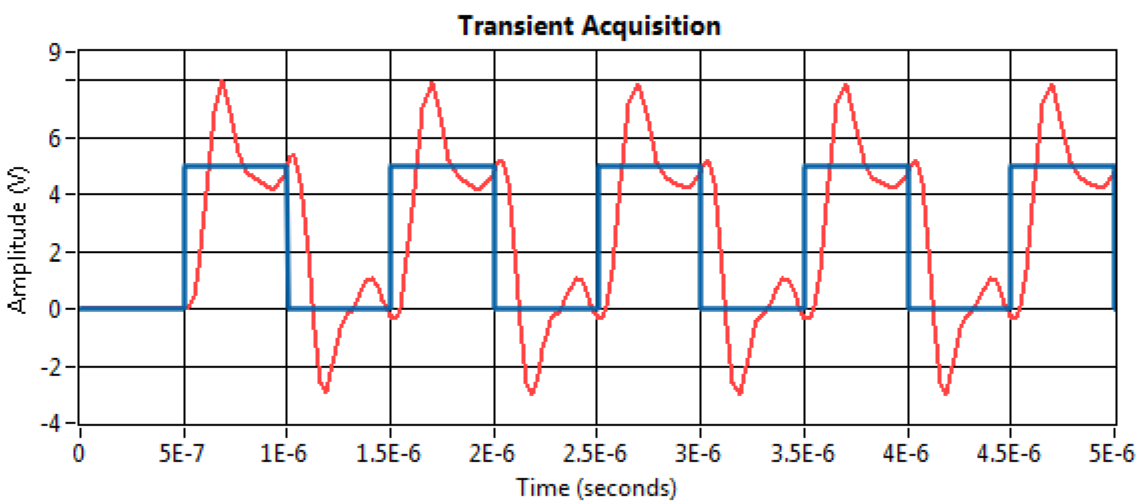
- ❑ Once you have finished reviewing the pre-built example VI, or have finished creating your own VI, proceed to the next step.

8. Run the VI.

- ❑ Click Run to start the VI.

Note If this is the first time you launch Multisim, this VI may take anywhere from 30 to seconds to 1 minute to run. It takes the same time as starting Multisim manually.

- ❑ Once the VI has completed, the XY Graph control will be populated with the analysis data. It will look similar to the following XY graph screen capture.



Caution Every time you run the Command Line VI, it will overwrite the files specified in the file paths if they already exist. To keep the files intact after each program iteration, add the program logic necessary to supply new file paths.

Summary

This tutorial explained the simplest process to get a text-based SPICE netlist and a SPICE nutmeg commands file to simulate from LabVIEW. This is particularly important when you want to have advanced access to the Multisim simulation engine for analyses.

Your LabVIEW programs will most likely be far more complex than the one presented here. However, the purpose of the tutorial is to show the simplest and most basic way to connect to the Multisim engine and simulate the SPICE commands.

If you want to review the pre-built example, open the SPICE Command Line VI located in the same folder as this tutorial. Remember that the two subVIs referenced in this tutorial were custom-made for this particular analysis.