

## Links

[Introduction](#)

[Using the QUCS](#)

[Creating Projects](#)

[Simulating  
Projects](#)

[DC Analysis](#)

[AC Analysis](#)

[Transient  
Response](#)

[PSpice Comparison  
to QUCS](#)

[Usability  
Comparison](#)

[Complexity  
Comparison](#)

[Conclusion](#)

[Bibliography](#)

# Quite Universal Circuit Simulator vs. PSpice

## Written By:

Brian Clearwaters

Adam Sheer

Ben White

## Revised By:

Mike Fanta

Dave Gamble

Sam Maher

Troy Shoemaker

## ABSTRACT

This white paper investigates the Quite Universal Circuit Simulator (QUCS). The program is an open source circuit simulator that is capable of doing simple analog simulations. The paper covers three of the program's basic simulations: DC analysis, AC analysis and transient response. QUCS is also compared to PSpice, a complex commercial circuit simulator. Usability and complexity will be directly compared assuming the user is completing a simple simulation. PSpice is an extremely detailed program, and allows the user to perform simulations on much more specific circuitry, while QUCS is a time saver that allows simulations of generic electronic circuits.

[Read More >](#)



QUCS

Quite Universal Circuit Simulator White Paper

April 28, 2006

## Links

[Introduction](#)

[Using the QUCS](#)

[Creating Projects](#)

[Simulating  
Projects](#)

[DC Analysis](#)

[AC Analysis](#)

[Transient  
Response](#)

[PSpice Comparison  
to QUCS](#)

[Usability  
Comparison](#)

[Complexity  
Comparison](#)

[Conclusion](#)

[Bibliography](#)

# INTRODUCTION

Electrical engineers use a type of design software called Electronic Design Automation or EDA. Engineers start by drawing a schematic of their circuit with the software they choose to use. Once a schematic is constructed, the EDA software can run simulations and show the circuit's performance (Moretti). EDA software began in the 1970's at Berkley with a program called SPICE (Simulation Program with Integrated Circuit Emphasis). Today many versions of SPICE exist, including PSpice which we discuss below. (Brorson)

The Quite Universal Circuit Simulator (QUCS) is a simple, open source EDA program. The program runs simulations much like commercially available packages such as PSpice, with generic components. QUCS is licensed under the General Public License (GPL), thus allowing the software to be used for free and changed by users. This program is also an ongoing project posted on SourceForge.net. Since the program is open source its source code can be modified and improved by anyone. Users can assist in debugging and upgrading the software. In this case, the program PSpice may be more suitable. This program allows several simulations with very specific components.

This paper will cover the three main simulations listed above in QUCS and make a comparison between QUCS and PSpice on the complexity and usability of the programs. This comparison will allow the user to make the correct decision as to which simulator may better suit their needs. This paper will give a brief overview of how to use the QUCS circuit simulator, and three main simulations (DC, AC, Transient) available in it.

[<](#) [1](#) [2](#) [3](#) [4](#) [5](#) [6](#) [7](#) [8](#) [9](#) [10](#) [11](#) [12](#) [>](#)



**QUCS**

Quite Universal Circuit Simulator White Paper

April 28, 2006

# Using the Quite Universal Circuit Simulator

## Links

[Introduction](#)

[Using the QUCS](#)

[Creating Projects](#)

[Simulating  
Projects](#)

[DC Analysis](#)

[AC Analysis](#)

[Transient  
Response](#)

[PSpice Comparison  
to QUCS](#)

[Usability  
Comparison](#)

[Complexity  
Comparison](#)

[Conclusion](#)

[Bibliography](#)

The QUCS program is available for many operating systems including Windows, Mac OS and Linux machines. The program has a graphical interface similar to other EDA software. The QUCS program is made to do analog simulations with generic components. The program has the ability to run several simulations including DC analysis, AC analysis and transient analysis. These analog simulations will give the designer vital information explaining the circuit's behavior. The DC simulation gives the DC voltage levels in the circuit. AC simulation is similar to DC but it gives AC voltages instead. Transient simulation measures the frequency variations of the circuit.

QUCS is a useful program for engineers and technicians of any level. The program is easy enough to use so that students can simulate their simple homework and lab circuits with it. While this program is easy to use, it is also very simple and may not provide adequate analysis required for a student or professional.

The first step to simulating a circuit is to create a new project. The circuit is then drawn so that it can be tested. QUCS is capable of doing a number of simulations and we will explain three common simulations below.

[<](#) [1](#) [2](#) **[3](#)** [4](#) [5](#) [6](#) [7](#) [8](#) [9](#) [10](#) [11](#) [12](#) [13](#) [>](#)



**QUCS**

Quite Universal Circuit Simulator White Paper

April 28, 2006

## Creating Projects

Upon opening QUCS, a *.qucs* directory is created. This is a main directory that is much like the main directory created in similar EDA programs. All files saved from this program are saved within this directory. Once QUCS is run for the first time, the *projects* tab is activated (Hertzog). When the user clicks *new project*, a project file is created, and all files associated with that project are saved within that project file. Once a new project is selected, it is loaded in the *main window*. This is the *project window*. The user may also open existing projects, as well as delete projects, from this *project tab* in the *work window*, on the left side of the screen. Figure 1, on page 3, gives an example of the *work window* and the *project window*.

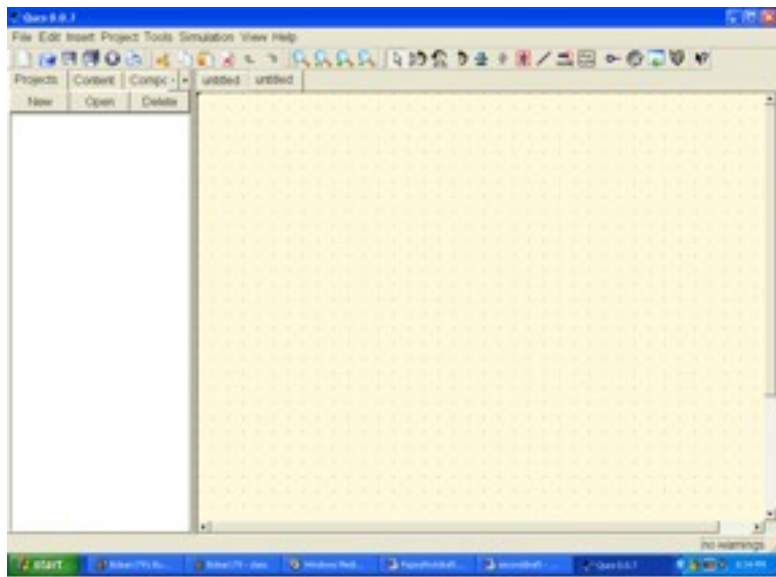


Figure 1 – QUCS main window

In the *work window* there are two other tabs along with the project tab. One is the *content tab*, which allows the user to view all current content for the open project, such as schematics, displays, etc. There is also a tab for *components*. From here the user can apply the components of the circuit to be simulated. The *components section* allows the user to choose general electrical components to place on the *project screen*. When the *components tab* is clicked, a drop down menu appears directly below it. This allows the user to choose the type of component (*linear*, *non-linear*, *sources*, etc.) that is to be placed in the circuit. The user may click on the type of component and drag the mouse over to the *project window*. The component is ready to be placed by clicking the mouse once the component is in the desired location on the *project screen*.

Values can be changed for each component by right clicking on the component and choosing the *edit properties* option. The components are connected by choosing the *wire* option in the toolbar option *insert* (the options for the *insert tab* and the *simulate tab* are also present in the toolbar). Once the *wire* option has been selected, the wire is placed by clicking on the open port of the component, signified by a red circle. Then the next click will be on another open port of a component, and this finishes the wire and connects the two components electrically. In order to simulate the circuit properly, the point of analysis on the circuit, or node, must be named. This is done by selecting the *wire* option on the toolbar, and further selecting *wire label*. Next, the wire at the point of analysis must be clicked to place the node and label at that point.



[Links](#)  
[Introduction](#)  
[Using the QUCS](#)  
[Creating Projects](#)  
[Simulating Projects](#)  
[DC Analysis](#)  
[AC Analysis](#)  
[Transient Response](#)  
[PSpice Comparison to QUCS](#)  
[Usability Comparison](#)  
[Complexity Comparison](#)  
[Conclusion](#)  
[Bibliography](#)

## Simulating Projects

### Links

[Introduction](#)

[Using the QUCS](#)

[Creating Projects](#)

[Simulating  
Projects](#)

[DC Analysis](#)

[AC Analysis](#)

[Transient  
Response](#)

[PSpice Comparison  
to QUCS](#)

[Usability  
Comparison](#)

[Complexity  
Comparison](#)

[Conclusion](#)

[Bibliography](#)

Once the circuit is complete with components, sources and node labels, the type of simulation can be chosen. This is done by clicking on simulations from the drop down menu under the components tab. DC, AC, transient, harmonic, s-parameter and parameter sweep are all options for simulations (Flucke). The type of simulation desired by the user is then placed on the project screen similar to the way components are placed, by being dragged onto the project window and then clicked to be put into place. Once the desired simulation is present on the project screen, the circuit is ready to be simulated. This is done by clicking simulate either on the toolbar or under the simulate pull-down tab. If there are errors with the circuit or the type of simulation chosen, a pop-up window will inform the user. If the simulation completes with no error, then the simulation window will appear over the project window. From here, the user can decide what kind of visual display they would like to observe. The type of display (Cartesian, Smith, and tabular charts) can then be chosen to view at the simulation window. These are chosen under the display option on the components tab. The choice of display will depend on what kind of circuit the user is simulating. If the circuit contains an AC source, the user will want to run an AC analysis, and so forth. Once the display type is placed on the project screen, a dialog box opens, allowing the user to choose what node to include in the chosen display. On the left side of this dialog box a list of nodes will be defined. Double clicking on the node of choice will put this node in the window on the right side of the dialog box. Once the correct node is placed in the display for analysis, the dialog box can be exited by clicking ok, and the output information for the node of analysis should be present in the project screen.

[<](#) [1](#) [2](#) [3](#) [4](#) **[5](#)** [6](#) [7](#) [8](#) [9](#) [10](#) [11](#) [12](#) [13](#) [>](#)



**QUCS**

Quite Universal Circuit Simulator White Paper

April 28, 2006

## DC Analysis

DC simulation is used to analyze the DC values of a circuit, such as a voltage bias. This can be achieved in QUCS by applying a DC source to passive components. Figure 2 below shows a schematic of a simple voltage divider circuit. Figure 3 then shows the output given by the DC simulation, in tabular format. Also, the point of analysis must be denoted, such as the node divide shown in Figure 2.

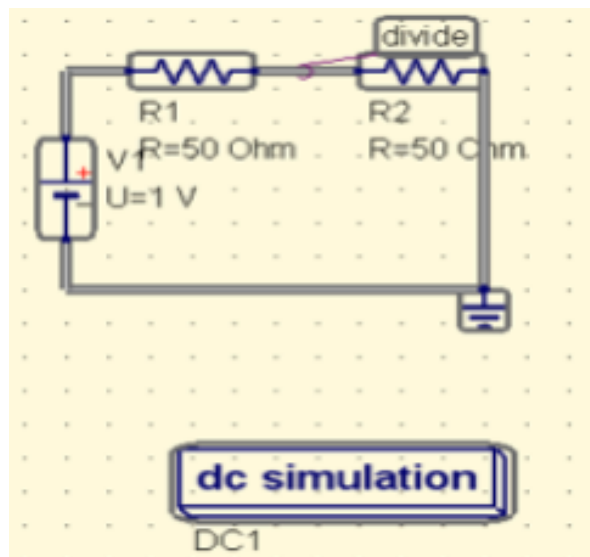


Figure 2 – Example DC simulation

number	divide.V
1	0.5

Figure 3 – Output voltage for DC simulation



### Links

[Introduction](#)

[Using the QUCS](#)

[Creating Projects](#)

[Simulating Projects](#)

[DC Analysis](#)

[AC Analysis](#)

[Transient Response](#)

[PSpice Comparison to QUCS](#)

[Usability Comparison](#)

[Complexity Comparison](#)

[Conclusion](#)

[Bibliography](#)

## Links

- Introduction
- Using the QUCS
- Creating Projects
- Simulating Projects
- DC Analysis
- AC Analysis
- Transient Response
- PSpice Comparison to QUCS
- Usability Comparison
- Complexity Comparison
- Conclusion
- Bibliography

# AC Analysis

AC simulation is used to observe AC voltages and currents. This is achieved similar to the DC simulation. An AC source must be applied to a circuit in order to analyze AC voltages and currents. Figure 4 demonstrates an AC signal amplifier that can be used to analyze AC parameters. Figure 5 demonstrates the output display given for the *tabular* option for the output voltage. The AC simulation box under the schematic must be included for the AC simulation for QUCS, unlike similar EDA programs. The output of the amplifier is labeled using the *wire label* option, in order to give the simulation a point of analysis.

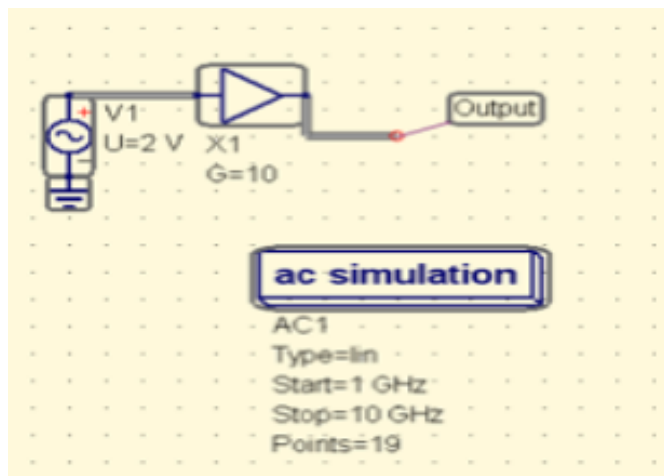


Figure 4 – Example AC Simulation

acfrequency	V1.i	Output.v
1e09	-0.04	40
1.5e09	-0.04	40
2e09	-0.04	40
2.5e09	-0.04	40
3e09	-0.04	40
3.5e09	-0.04	40
4e09	-0.04	40

Figure 5 – Output Voltage for AC



## Links

[Introduction](#)

[Using the QUCS](#)

[Creating Projects](#)

[Simulating  
Projects](#)

[DC Analysis](#)

[AC Analysis](#)

[Transient  
Response](#)

[PSpice Comparison  
to QUCS](#)

[Usability  
Comparison](#)

[Complexity  
Comparison](#)

[Conclusion](#)

[Bibliography](#)

# Transient Response

The transient simulation is available in order to analyze the circuits with frequency dependant components. This can be used to view the amplitudes and times of harmonics, or voltage impulses over a period of time. The circuit must be constructed in the *project window*, and the *transient simulation* box must be dragged into that window. The circuit can then be simulated provided that there are no errors in the construction of the circuit. The output can then be viewed as a tabular chart or a graph, which will give a time-domain analysis. This may be used to view amplitudes of voltage spikes, or times of frequency harmonics.

[<](#) [1](#) [2](#) [3](#) [4](#) [5](#) [6](#) [7](#) **8** [9](#) [10](#) [11](#) [12](#) [13](#) [>](#)



QUCS

Quite Universal Circuit Simulator White Paper

April 28, 2006



## PSpice Comparison to QUCS

PSpice is a commercially available EDA program that is similar to QUCS. PSpice has most of the options that QUCS has and some additional features. PSpice is widely used throughout the engineering, technology, and academic communities. It is the standard to which other EDA programs are measured. PSpice has the ability to run the same analyses that QUCS can and can also give more in depth information. The following comparison covers the usability and complexity of both programs.

### Links

[Introduction](#)

[Using the QUCS](#)

[Creating Projects](#)

[Simulating  
Projects](#)

[DC Analysis](#)

[AC Analysis](#)

[Transient  
Response](#)

[PSpice Comparison  
to QUCS](#)

[Usability  
Comparison](#)

[Complexity  
Comparison](#)

[Conclusion](#)

[Bibliography](#)

[<](#) [1](#) [2](#) [3](#) [4](#) [5](#) [6](#) [7](#) [8](#) **[9](#)** [10](#) [11](#) [12](#) [13](#) [>](#)



**QUCS**

Quite Universal Circuit Simulator White Paper

April 28, 2006

## Links

- Introduction
- Using the QUCS
- Creating Projects
- Simulating Projects
- DC Analysis
- AC Analysis
- Transient Response
- PSpice Comparison to QUCS
- Usability Comparison
- Complexity Comparison
- Conclusion
- Bibliography

# Usability Comparison

QUCS uses generic components which makes the construction of a circuit fairly simple. However, PSpice allows the user to select specific components with industry standard part numbers and specifications. Searching for these components can take up more of the user's time when constructing the circuit, which is a disadvantage of PSpice. Placing the components in the project window and making electrical connections is similar between both programs. Both programs allow the user to draw wires between components by selecting a wire option and clicking on the start and end position of the wire.

Both programs also allow the user to view the result of the simulation as a graph or chart. PSpice allows multiple plots to be viewed simultaneously, such as voltage, power, etc. The results of a QUCS simulation are much more limited. Only one parameter can be viewed at a time. Also, specific points, such as a voltage at a certain time, can be selected and marked on the output plot in PSpice, an option that is not available in QUCS. An example of an output plot containing markers is shown in Figure 6.

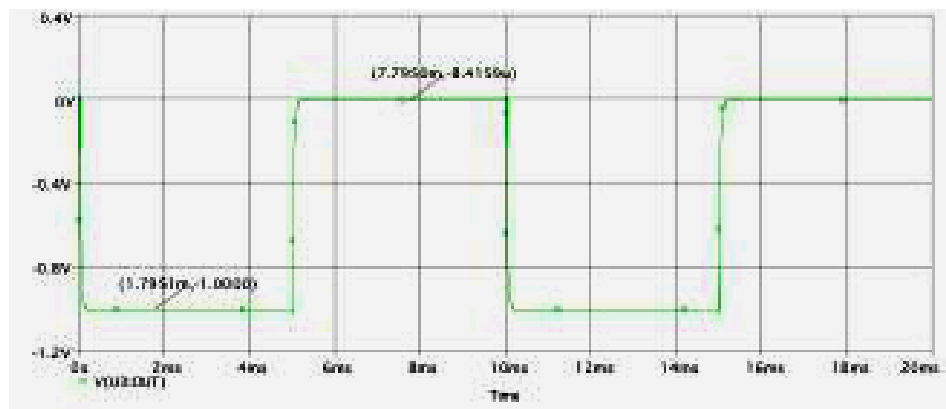


Figure 6 – PSpice output plot containing markers (in grey)

The ease of use of these programs is dependent on the user's familiarity with circuit simulator programs. For a beginner using a circuit simulator for the first time, QUCS will save a considerable amount of the user's time. It is also ideal for simple circuit simulations, which would be simulated by a beginner. For a user with circuit simulation experience, PSpice is a more powerful application. The more experienced user will likely use the program for more complex circuitry as well as detailed output results.



## Links

[Introduction](#)

[Using the QUCS](#)

[Creating Projects](#)

[Simulating  
Projects](#)

[DC Analysis](#)

[AC Analysis](#)

[Transient  
Response](#)

[PSpice Comparison  
to QUCS](#)

[Usability  
Comparison](#)

[Complexity  
Comparison](#)

[Conclusion](#)

[Bibliography](#)

## Complexity Comparison

PSpice contains libraries full of specific components with manufacturer specifications. These components are included so the user may obtain realistic simulation results. QUCS contains generic parts, which gives the user ideal simulation results, which may not exhibit a realistic output.

Components can also be modified in PSpice. The specific electrical properties of the components can be changed to meet user needs.

The only modifications that can be made to components using QUCS are component values. The models for components in QUCS are ideal, where the models for components in PSpice are more realistic. Figure 7 below shows an example of a circuit schematic in PSpice. Note the part number, LF411, on the operational amplifier. This is an example of how PSpice allows the user to select specific components for simulation.

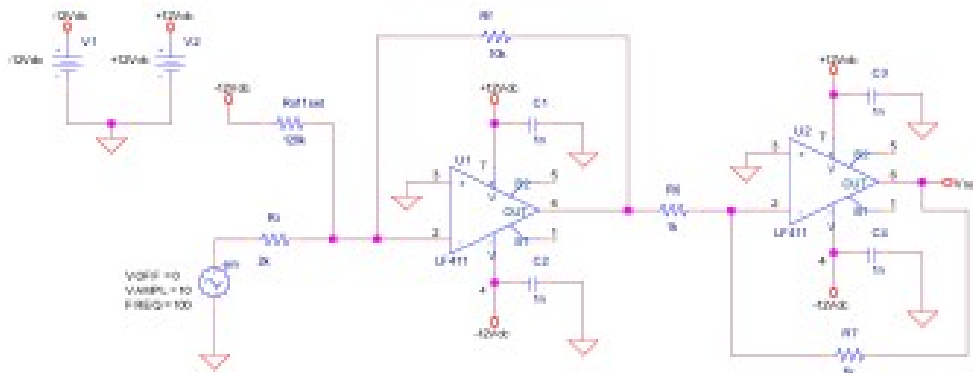


Figure 7 – Circuit schematic utilizing PSpice

PSpice is a much more complex circuit simulator than QUCS. With the ability to use real components, PSpice can give the user more realistic results for a circuit that could be constructed and tested. QUCS is not a complex program and will only provide the user with ideal results for simple circuitry

## Links

[Introduction](#)

[Using the QUCS](#)

[Creating Projects](#)

[Simulating  
Projects](#)

[DC Analysis](#)

[AC Analysis](#)

[Transient  
Response](#)

[PSpice Comparison  
to QUCS](#)

[Usability  
Comparison](#)

[Complexity  
Comparison](#)

[Conclusion](#)

[Bibliography](#)

## Conclusion

The Quite Universal Circuit Simulator is a great example of open source software. The program is useful for simple analog circuit simulation. The simple interface and components allow it to be used by students. The simulations are detailed and can be used by engineers for complex design. QUCS provides a simple alternative to the complex and expensive commercial alternatives.

[<](#) [1](#) [2](#) [3](#) [4](#) [5](#) [6](#) [7](#) [8](#) [9](#) [10](#) [11](#) **[12](#)** [13](#) [>](#)



**QUCS**

Quite Universal Circuit Simulator White Paper

April 28, 2006

## BIBLIOGRAPHY

- Andresen, Richard P. "White Paper." 5Spice White Paper. 2004. 5Spice. 13 Oct. 2005 <<http://www.5spice.com/whtpaper.htm>>.
- "Basic PSpice Information." Basic PSpice Information. 19 Jan 1995. Duke University. 12 Oct. 2005 <<http://www.ee.duke.edu/~cec/final/node26.html>>.
- Brorson, Stuart. "Circuit simulation using gEDA and SPICE -- HOWTO." Circuit simulation using gEDA and SPICE -- HOWTO. 2004. Electroniscript, inc.. 13 Oct. 2005 <<http://www.brorson.com/gEDA/SPICE/intro.html>>.
- Flucke, Jens. Qucs project. 2004. SourceForge.net 13 Oct. 2005 <>.  
Hertzog, Raphaë'91l. Overview of qucs source package. Debian Package Tracking System . 13 Oct. 2005 <<http://packages.qa.debian.org/q/qucs.html>>.
- Jahn , Stefan. QUCS Technical Papers. 27 Jul 2005. SourceForge.net 13 Oct. 2005 <<http://qucs.sourceforge.net/tech/technical.html>>.
- Johnson, Adam . PSpice Tutorial. 25 Nov 2000. 11 Wilfrid Laurier University Oct. 2005 <[http://denethor.wlu.ca/pc300/PSpice/pspice\\_tutorial.html](http://denethor.wlu.ca/pc300/PSpice/pspice_tutorial.html)>.
- Moretti, Gabe. "EDA open systems: 'Open' is not a magic word." EDN PDF. 15 February 2001. EDN Magazine. 13 Oct. 2005 <<http://www.edn.com/contents/images/63734.pdf>>.
- Project: Quite Universal Circuit Simulator: Summary. SourceForge.net. 13 Oct. 2005 <<http://sourceforge.net/projects/qucs>>.
- "PSpice Resources." Orcad Community. 2004. Orcad. 13 Oct. 2005 <<http://www.orcad.com/community.pspice.an.aspx>>.
- "PSpice Tutorial." Using MOS Transistors In Orcad 9.1. 18 Oct 2000. Claris Home Page. 12 Oct. 2005 <<http://www.icaen.uiowa.edu/~sparks/SPICEMODELS.htm>>.
- "QUCS." Math & Science. Apple Computer, Inc. 13 Oct. 2005 <[http://www.apple.com/downloads/macosx/math\\_science/qucs.html](http://www.apple.com/downloads/macosx/math_science/qucs.html)>.
- Andresen, Richard P. "White Paper." 5Spice White Paper. 2004. 5Spice. 13 Oct. 2005 <<http://www.5spice.com/whtpaper.htm>>.
- "Basic PSpice Information." Basic PSpice Information. 19 Jan 1995. Duke University. 12 Oct. 2005 <<http://www.ee.duke.edu/~cec/final/node26.html>>.
- Brorson, Stuart. "Circuit simulation using gEDA and SPICE -- HOWTO." Circuit simulation using gEDA and SPICE -- HOWTO. 2004. Electroniscript, inc.. 13 Oct. 2005 <<http://www.brorson.com/gEDA/SPICE/intro.html>>.
- Flucke, Jens. Qucs project. 2004. SourceForge.net 13 Oct. 2005 <>.  
Hertzog, Raphaë'91l. Overview of qucs source package. Debian Package Tracking System . 13 Oct. 2005 <<http://packages.qa.debian.org/q/qucs.html>>.
- Jahn , Stefan. QUCS Technical Papers. 27 Jul 2005. SourceForge.net 13 Oct. 2005 <<http://qucs.sourceforge.net/tech/technical.html>>.
- Johnson, Adam . PSpice Tutorial. 25 Nov 2000. 11 Wilfrid Laurier University Oct. 2005 <[http://denethor.wlu.ca/pc300/PSpice/pspice\\_tutorial.html](http://denethor.wlu.ca/pc300/PSpice/pspice_tutorial.html)>.
- Moretti, Gabe. "EDA open systems: 'Open' is not a magic word." EDN PDF. 15 February 2001. EDN Magazine. 13 Oct. 2005 <<http://www.edn.com/contents/images/63734.pdf>>.
- Project: Quite Universal Circuit Simulator: Summary. SourceForge.net. 13 Oct. 2005 <<http://sourceforge.net/projects/qucs>>.
- "PSpice Resources." Orcad Community. 2004. Orcad. 13 Oct. 2005 <<http://www.orcad.com/community.pspice.an.aspx>>.
- "PSpice Tutorial." Using MOS Transistors In Orcad 9.1. 18 Oct 2000. Claris Home Page. 12 Oct. 2005 <<http://www.icaen.uiowa.edu/~sparks/SPICEMODELS.htm>>.
- "QUCS." Math & Science. Apple Computer, Inc. 13 Oct. 2005 <[http://www.apple.com/downloads/macosx/math\\_science/qucs.html](http://www.apple.com/downloads/macosx/math_science/qucs.html)>.

[<](#)
[1](#)
[2](#)
[3](#)
[4](#)
[5](#)
[6](#)
[7](#)
[8](#)
[9](#)
[10](#)
[11](#)
[12](#)
[13](#)
[>](#)

	<b>QUCS</b>	Quite Universal Circuit Simulator White Paper
		April 28, 2006

### Links

Introduction

Using the QUCS

Creating Projects

Simulating Projects

DC Analysis

AC Analysis

Transient Response

PSpice Comparison to QUCS

Usability Comparison

Complexity Comparison

Conclusion

Bibliography