



RF and mmWave Circuit Design

QUCS-S INSTALLATION

dr. Carlos Mendes Jr
prof. dr.-ir. Peter Baltus
prof. dr. Marion Matters

Department of Electrical Engineering, Integrated Circuits Group

coursera

TU/e

Download of Qucs-S and NGSpice

Windows

1. [Download Qucs-S \(Qucs with Spice\)](#)
 - [Qucs-S Help](#)
2. [Download NGSpice for Qucs-S](#)
 - OBS.: Unpacking **has** to go to C:\SPICE

Installation of Qucs-S

Qucs-S: Qucs with SPICE

Download links

The latest stable release is Qucs-0.0.22. It is based on stable Qucs-0.0.19:

- Documentation at the readthedocs.io and [here](#)
- Source tarball: [qucs-s-0.0.22.tar.gz](#)
- [Github repository](#)
- AppImage for all Linux distributions [at Github](#)
- Debian repository (32 and 64 bit), built with openSUSE OBS: [Debian 10 "Buster"](#), [Debian 9 "Stretch"](#), [Debian 8 "Jessie"](#), [Debian 7 "Wheezy"](#) and Ubuntu [14.04](#), [16.04](#), and [18.04](#)
- RPM Packages (32 and 64 bit) for [CentOS](#) and [Fedora-24, 25, and 26](#)
- Slackware >=14.1 [SlackBuild](#)
- Windows installer (Zipped EXE) [qucs-s-0.0.22-setup.zip](#)

[\(Installation instructions...\)](#)

Contribution guide

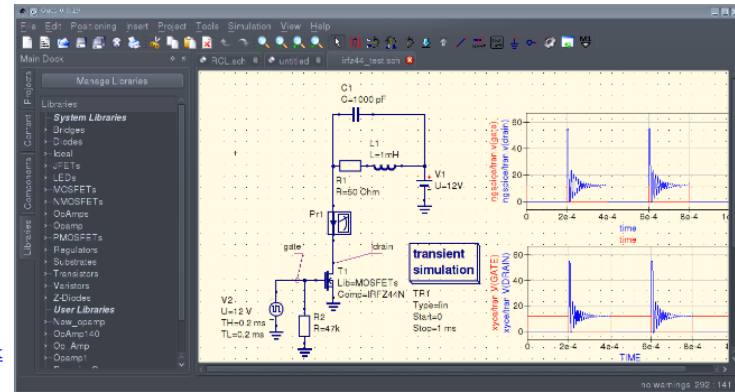
Qucs-S is open for everyone's contribution. See [here](#) for contribution guide.

[Click here to download](#)

News

- *January, 19, 2020* Qucs-S-0.0.22 is released! See [Release notes and download link](#)
- *October, 31, 2018* Qucs-S-0.0.21 is released! See [Release notes and download link](#)
- *June, 24, 2018* Added packages for Ubuntu 18.04
- *October, 31, 2017* Qucs-S-0.0.20 is released! See [Release notes and download link](#)
- *October, 25, 2017* Added packages for CentOS and Fedora
- *January, 26, 2017* Qucs-S 0.0.19 is released! The first stable release. [Release announcement](#)
- *November 8, 2016* Qucs-S RC8 released. [Release notes and download link](#)
- *September 3, 2016* Qucs-S RC7 released. [Release notes and download link](#)
- *May 15, 2016* Qucs-S RC6 released. [Release notes and download link](#)
- *March 23, 2016* Qucs-S RC5 released. [Release notes and download link](#)
- *January 31, 2016* Qucs-S RC4 released. [Release notes and download link](#)
- *August 29, 2015* Qucs-S RC3 released.
- *July 28, 2015* Qucs-S RC2 released.
- *July 25, 2015* Qucs-S RC1 released.

Simulation example with Qucs-S and Ngspice

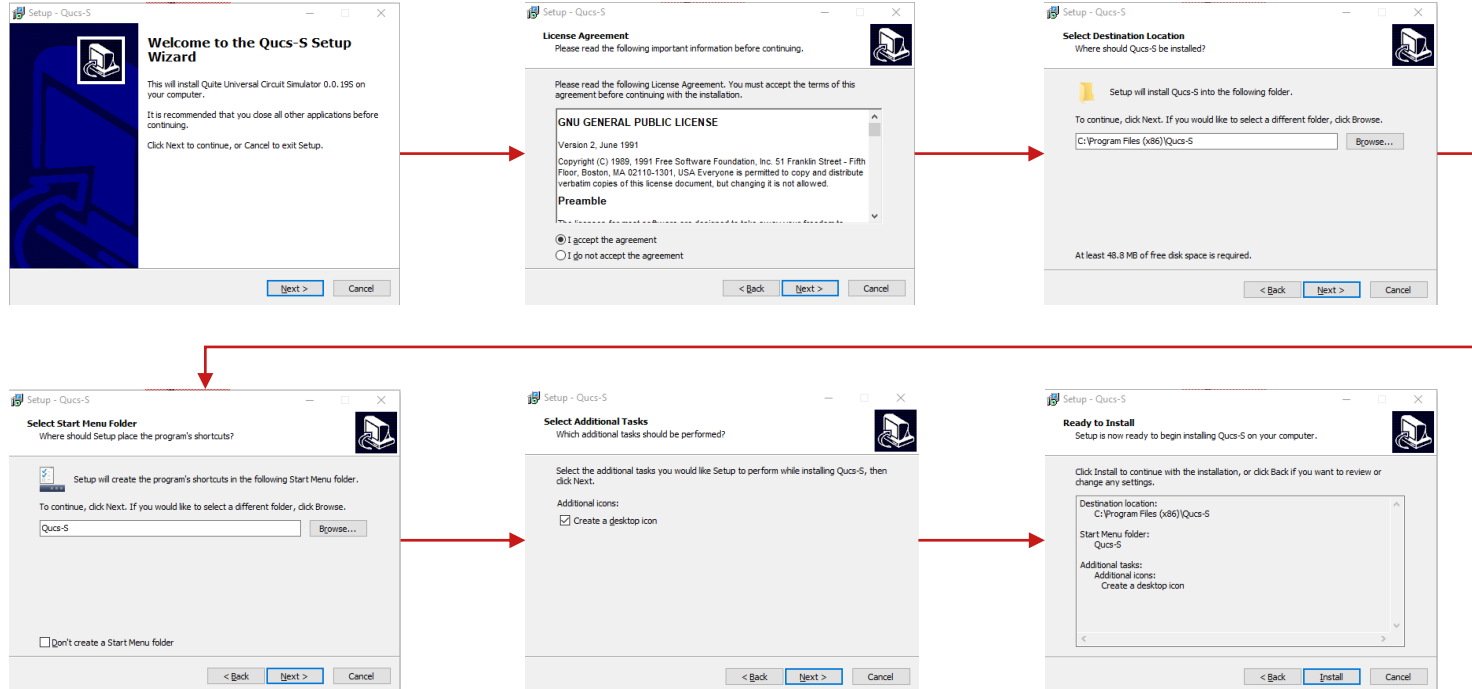


[\(More screenshots...\)](#)

Publications

Qucs-S is also a research software. Check our [publication list](#).

Installation of Qucs-S



Installation of NGSpice for Qucs-S

Qucs-S: Qucs with SPICE

Download links

The latest stable release is Qucs-S 0.0.22. It is based on stable Qucs-S 0.0.19:

- Documentation at the [readthedocs.io](#) and [here](#)
- Source tarball: [qucs-s-0.0.22.tar.gz](#)
- [Github repository](#)
- Appliance for all Linux distributions [at Github](#)
- Debian repository (32 and 64 bit), built with openSUSE OBS: [Debian 10 "Buster"](#), [Debian 9 "Stretch"](#), [Debian 8 "Jessie"](#), [Debian 7 "Wheezy"](#) and [Ubuntu 14.04](#), [16.04](#), and [18.04](#)
- RPM Packages (32 and 64 bit) for [CentOS](#) and [Fedora-24](#), [25](#), and [26](#)
- Slackware>=14.1 [SlackBuild](#)
- Windows installer (Zipped EXE): [qucs-s-0.0.22-setup.zip](#)

[\(Installation instructions...\)](#)

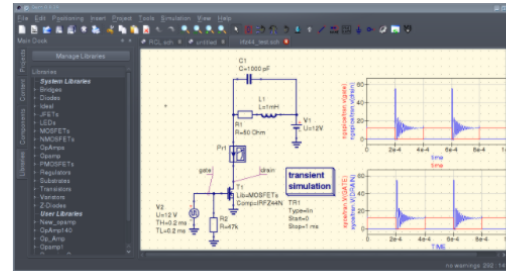
Contribution guide

Qucs-S is open for everyone's contribution. See [here](#) for contribution guide.

News

- *January, 19, 2020* Qucs-S-0.0.22 is released! See [Release notes and download link](#)
- *October, 31, 2018* Qucs-S-0.0.21 is released! See [Release notes and download link](#)
- *June, 24, 2018* Added packages for Ubuntu 18.04
- *October, 31, 2017* Qucs-S-0.0.20 is released! See [Release notes and download link](#)
- *October, 25, 2017* Added packages for CentOS and Fedora
- *January, 26, 2017* Qucs-S 0.0.19 is released! The first stable release. [Release announcement](#)
- *November 8, 2016* Qucs-S RC8 released. [Release notes and download link](#)
- *September 3, 2016* Qucs-S RC7 released. [Release notes and download link](#)
- *May 15, 2016* Qucs-S RC6 released. [Release notes and download link](#)
- *March 23, 2016* Qucs-S RC5 released. [Release notes and download link](#)
- *January 31, 2016* Qucs-S RC4 released. [Release notes and download link](#)
- *August 29, 2015* Qucs-S RC3 released.
- *July 28, 2015* Qucs-S RC2 released.
- *July 25, 2015* Qucs-S RC1 released.

Simulation example with Qucs-S and Ngspice



[\(More screenshots...\)](#)

Publications

Qucs-S is also a research software. Check our [publication list](#).

What is Qucs-S?

Qucs-S is a spin-off of the [Qucs](#) cross-platform circuit simulator. "S" letter indicates SPICE. The purpose of the Qucs-S subproject is to use free SPICE circuit simulation kernels with the Qucs GUI. It merges the power of SPICE and the simplicity of the Qucs GUI. Qucs intentionally uses its own SPICE incompatible simulation kernel Qucsator. It has advanced RF and AC domain simulation features, but most of the existing industrial SPICE models are incompatible with it. Qucs-S is not a simulator by itself, but it requires to use a simulation backend with it. The schematic document format of Qucs and Qucs-S are fully compatible. Qucs-S allows to use the following simulation kernels with it:

- **Ngspice** is recommended to use. Ngspice is powerful mixed-level/mixed-signal circuit simulator. The most of industrial SPICE models are compatible with Ngspice. It has an excellent performance for time-domain simulation of switching circuits and powerful postprocessor.
- **XYCE** is a new SPICE-compatible circuit simulator written by Sandia from the scratch. It supports basic SPICE simulation types and has an advanced RF simulation features such as Harmonic balance simulation.
- **SpiceOpus** is developed by the Faculty of Electrical Engineering of the Ljubljana University. It based on the SPICE-3f5 code
- Qucsator as backward compatible

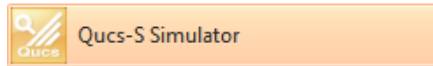
Click here to be forwarded

Installation of Qucs-S

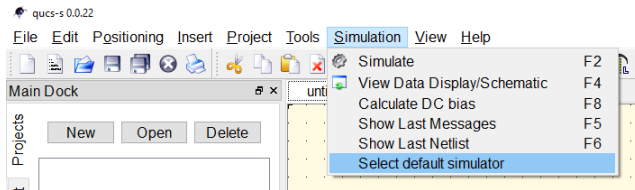
Unzip Ngspice package strictly to the C:\SPICE location.

Setting NGSpice into Qucs-S

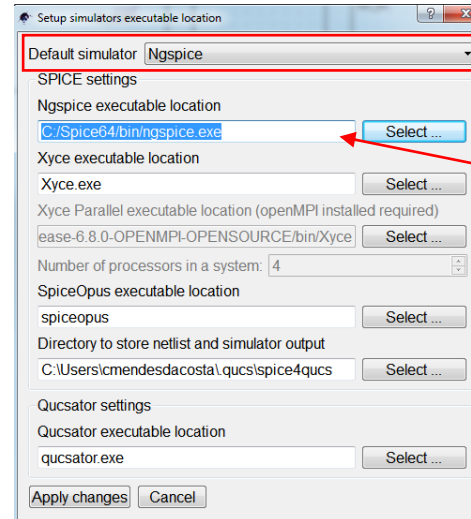
1. From *Start* menu open Qucs-S Simulator



2. Go to *Simulation* menu and open *Select default simulator*



3. Select Ngspice in the Default simulator, and set its path to C:\SPICE

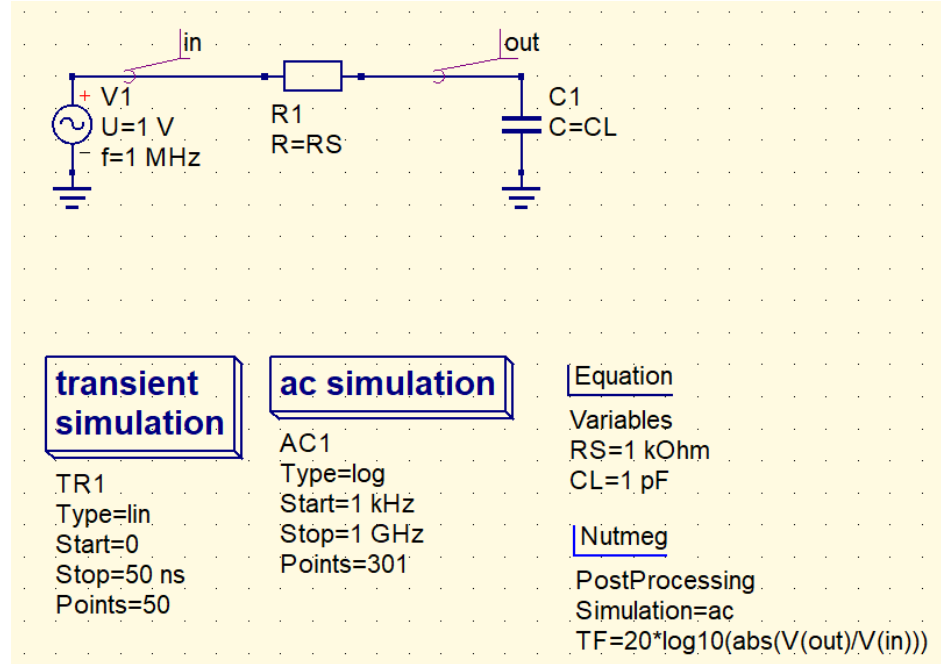


Path where
you saved the
NGSpice. It has
to be C:\SPICE

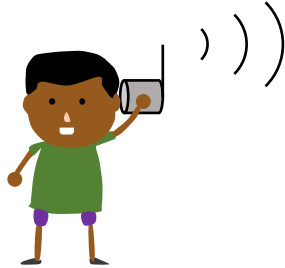
Simulation example

Low Pass Filter Simulation

- Transient
- AC



Thanks for watching!



C.A.M.Costa.Junior@tue.nl

P.G.M.Baltus@tue.nl

