



---

# INTRODUCTION TO PSPICE

---

Rahul Behl



MAY 18, 2014

---

## Preface

Introduction to PSpice consists of a set of tutorials which would guide the user to simulate basic analog and digital circuits using PSpice. The tutorials would provide step by step guide from installing PSpice to modeling and simulating circuits on PSpice. I hope you learn something valuable from these tutorials and get a better understanding of how different circuits work and what the characteristics of these circuits. For any other information feel free to contact me on [raulbehl@gmail.com](mailto:raulbehl@gmail.com). I'll be happy to help.

Happy Coding!

---

# Tutorial 1

## *Downloading PSpice*

Welcome to your first tutorial on PSpice. After some basic introduction to Pspice this tutorial would guide towards the installation and setting up the environment for PSpice.

SPICE (Simulation Program with Integrated Circuit Emphasis) is a general-purpose, open source analog electronic circuit simulator. It is a powerful program that is used in integrated circuit and board-level design to check the integrity of circuit designs and to predict circuit behavior. PSpice is a SPICE analog circuit and digital logic simulation program for Microsoft Windows. The name is an acronym for Personal SPICE.

### Downloading PSpice:

Since this tutorial series is concerned with PSpice only thus, the downloading procedure are discussed only for Windows (excluding OSX, Linux based kernels). Since PSpice is a freely available software you can easily download it from the internet. Once the file is downloaded install all the required files to get started with PSpice.

Note – Since the installation procedure is quite trivial the step wise information is omitted from his tutorial. If you face any problems feel free to contact the author.

Once the installation is complete, normally PSpice would open up and you'll see a screen similar to the one shown in figure below



Figure 1 – Opening Screen PSpice

If you reached this screen, congratulations you have successfully installed PSpice and now you are ready to code some simple (and complex as we proceed) analog and digital circuits.