

For today's lab, let's do a walkthrough with Altium Designer.

🏠 > Altium Designer Documentation >

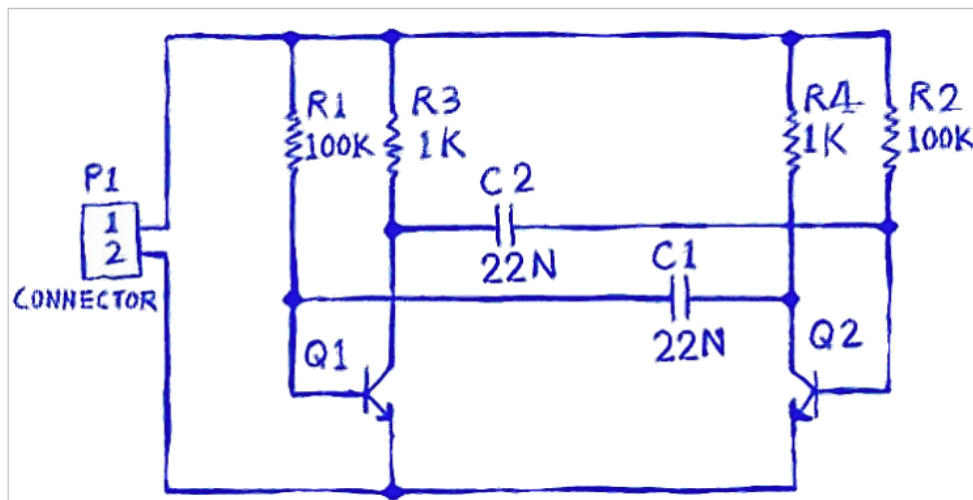
# Tutorial - A Complete Design Walkthrough with Altium Designer

Created: November 19, 2024 | Updated: October 01, 2025

Welcome to the world of electronic product development in Altium's world-class electronic design software. This tutorial will help you get started by taking you through the entire process of designing a simple PCB – from idea to output files.

## The Design

The design for which you will be creating the schematic and designing a printed circuit board (PCB) is a simple astable multivibrator. The circuit is shown below; it uses two general-purpose NPN transistors configured as a self-running astable multivibrator.



Circuit for the multivibrator

Here's the link:

<https://www.altium.com/documentation/altium-designer/tutorial>

At the end of the tutorial, you will finish the entire design process in Altium Designer: from a blank schematic sheet to a finished PCB. Have fun!