

1. Project Creation

Introduction

This page includes a quick overview about how to use the PCB Design Software - Altium. You'll learn the basic process on how Printed Circuit Boards (PCBs) are made by using the tools in Altium.

For additional information on learning the Altium basics, please access the tutorial page on their website: [Altium Tutorials](#)

Goals

- **Learning**
 - How are Printed Circuit Board Made?
 - What is Altium and how do you use it?
- **Practical**
 - Creating a circuit schematic
 - Developing a board layout and conducting a Design Rule Check
 - Preparing a board design for manufacturing

“ This tutorial is a Work in Progress!

Before starting your project, make sure you created are added to the Sooner Competitive Robotics Altium Workspace and have an account created.

PCB Basics

Information on:

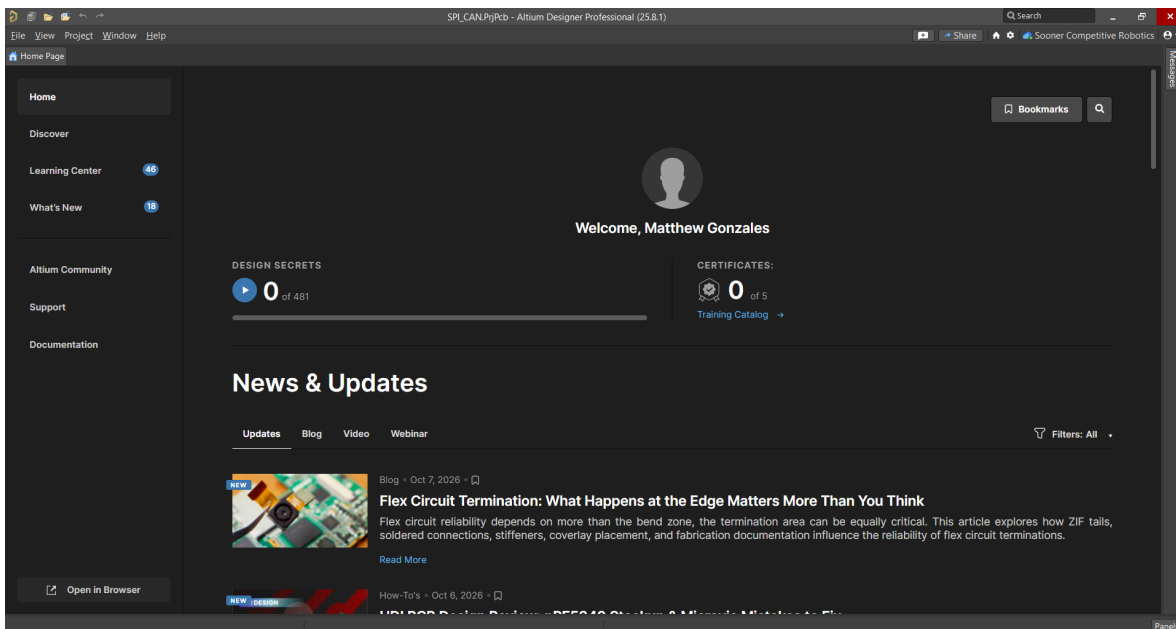
- What is a PCB
- What is Altium

Project Overview - SPI-CAN Breakout Board

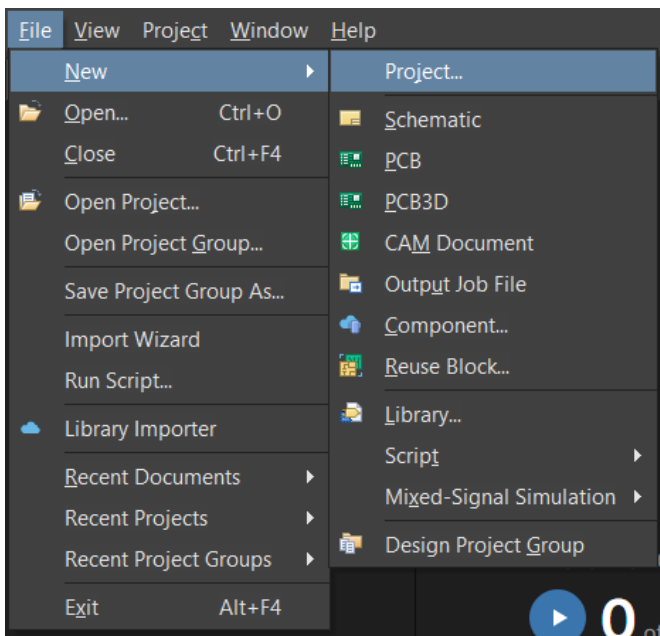
Insert Information Here...

Creating a Project

To start, open the Altium application on your device. Upon Altium loading, you should be greeted by a home page that looks similar to below:

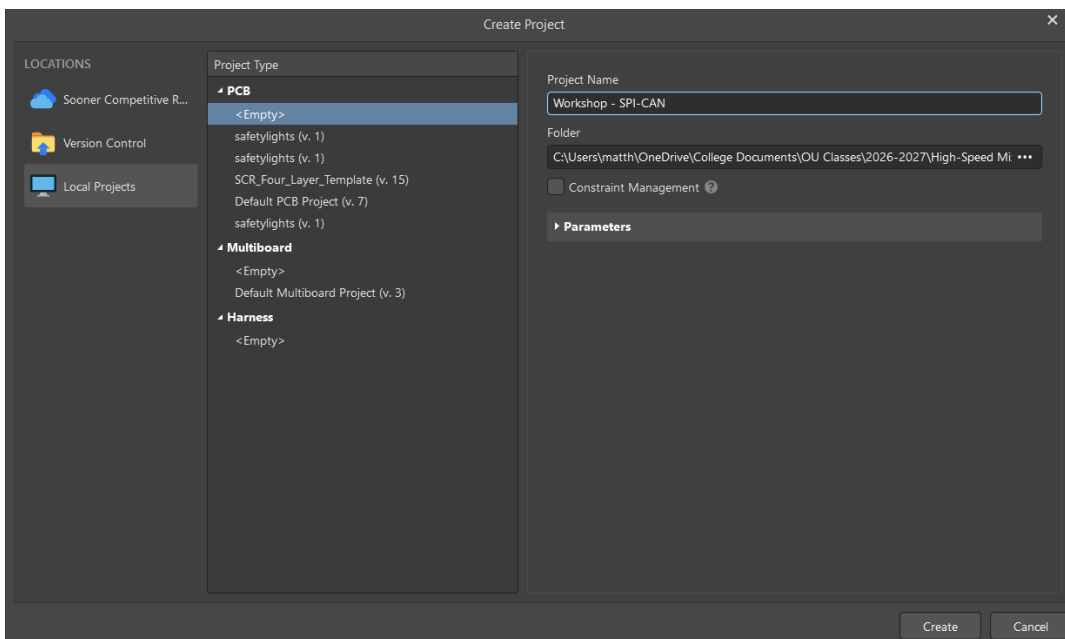


To start a new project, navigate to the top-left toolbar and select **File** --> select **New** --> then select **Project**



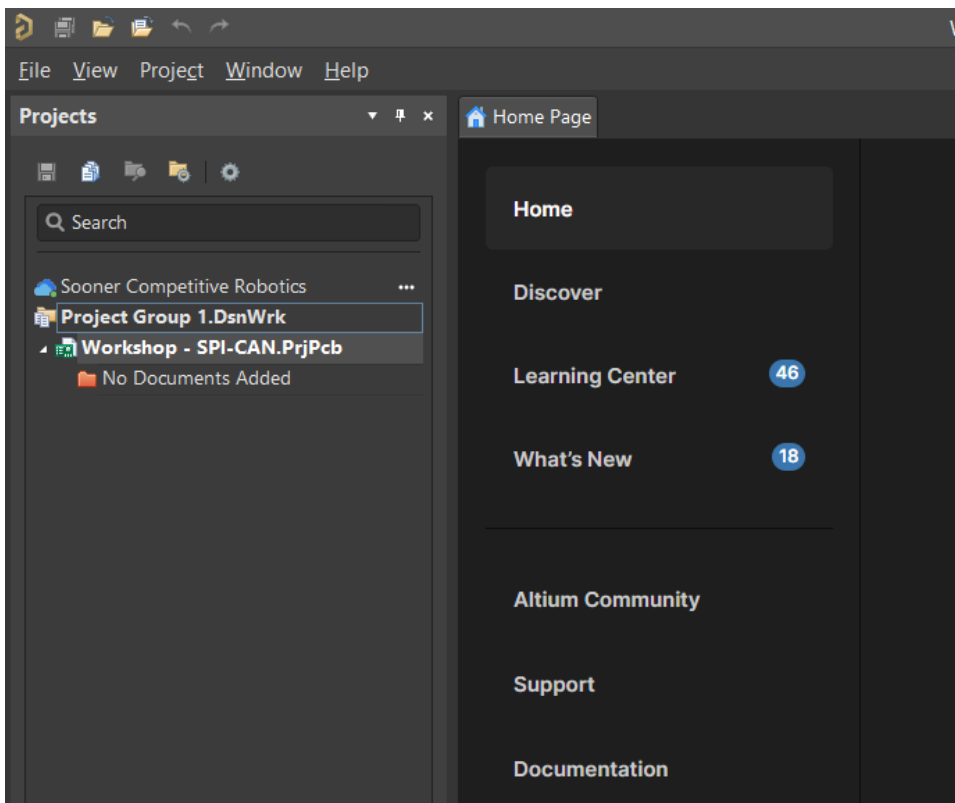
In the new window under **Project Name**, name your project "*Workshop - SPI-CAN*"

- Under **Folder**, make sure your project is saved in a file location you can find. You can change the file location by selecting the three dots "..." on the right.

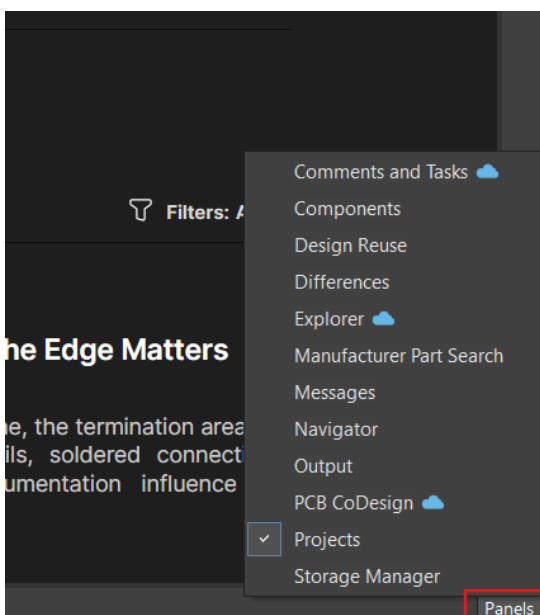


Upon creating a new project, the **Projects Panel** will now open, where you can view all projects you have open.

- In addition, your project's repository will be saved in the Sooner Competitive Robotics workspace.

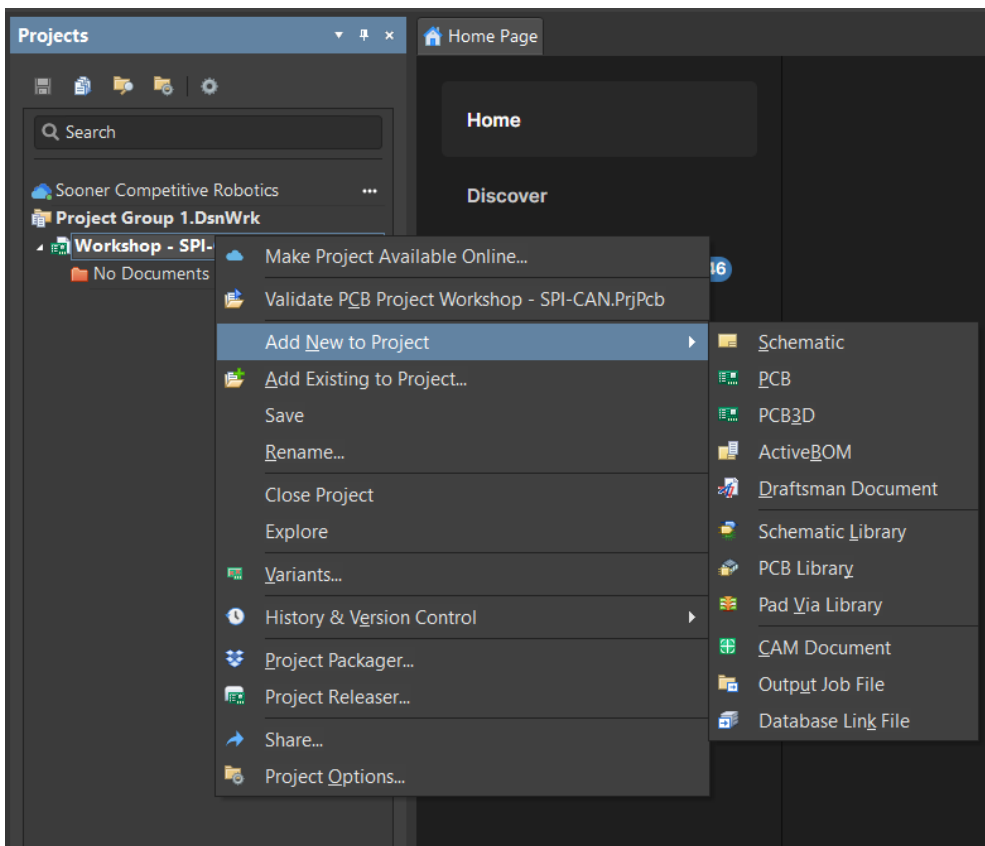


On the bottom right of your window, you'll find the **Panels** toggle. This toggle stores a ton of tools and will come in handy later. For now, just know you can reopen the Projects panel through accessing this toggle.



Adding to a Project

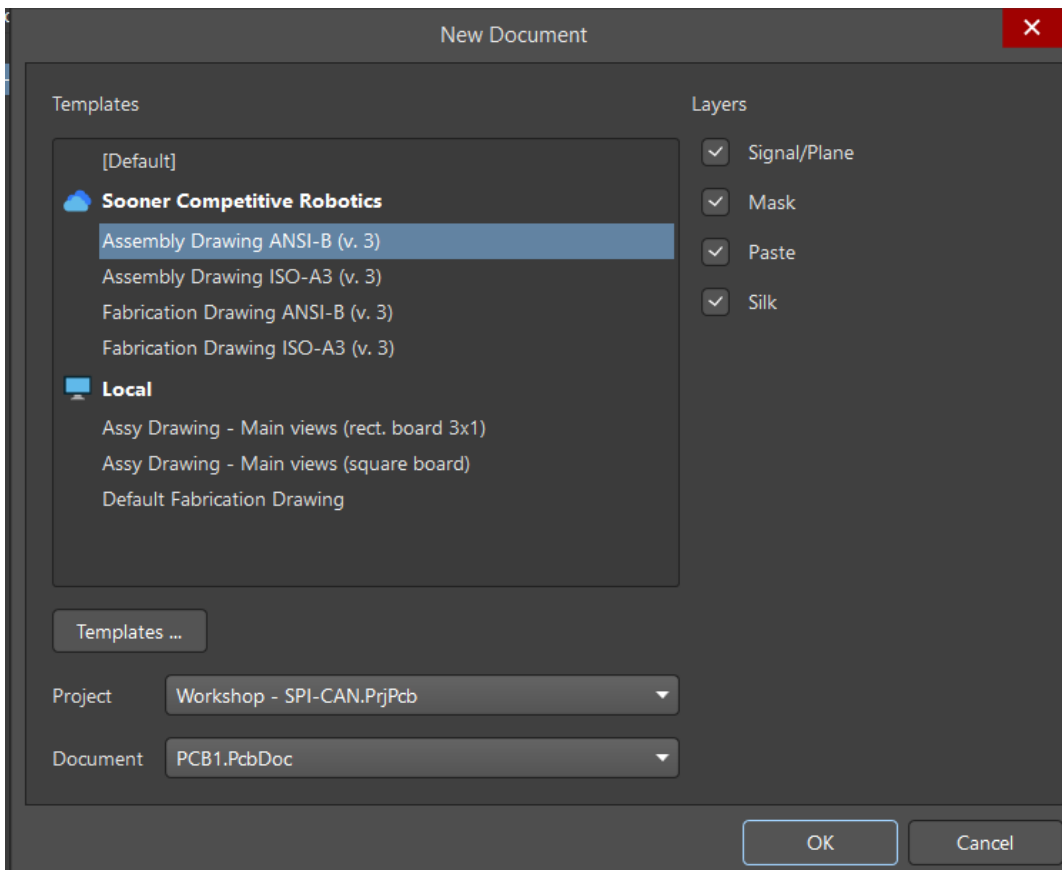
Now, your project is currently empty. To get started, you'll need to add documents by **right-clicking** on the "**Workshop - SPI-CAN.PrjPcb**" icon. In the new window, hover over **Add New to Project**, then you can add any documents you need.



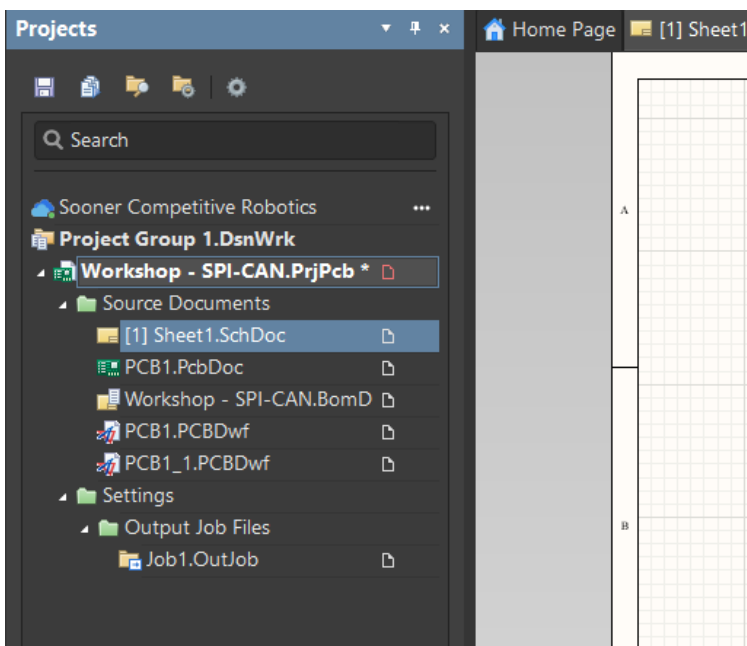
Go ahead and add the following documents to your project:

- Schematic
- PCB
- ActiveBOM
- Draftsman Document
 - Assembly Drawing
 - Fabrication Drawing
- Output Job File

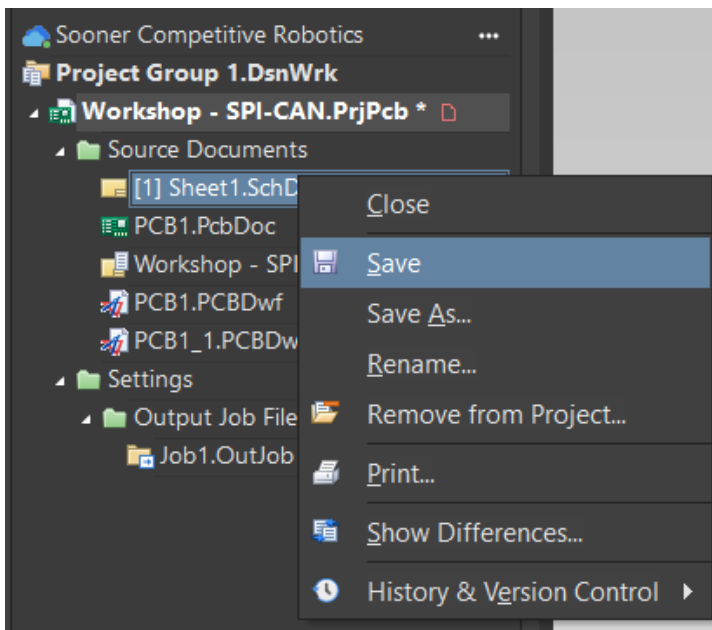
For the Draftsman Documents, a new window will appear. For the Assembly Drawing, select **Assembly Drawing ANSI-B (v. 3)**, then press **OK**. For the Fabrication Drawing, select **Fabrication Drawing ANSI-B (v. 3)**, then press **OK**.



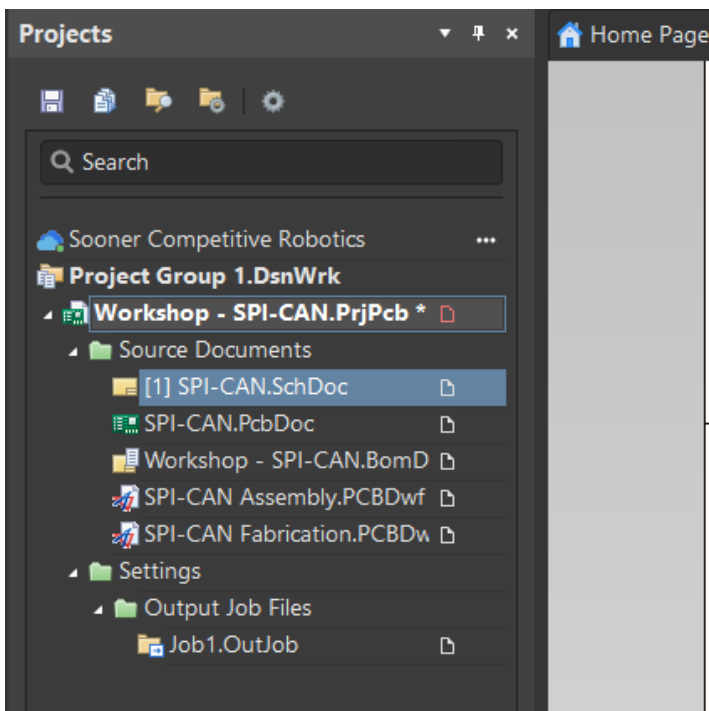
After you add the required documents to your project, your Projects panel should look like the screenshot below:



Before you start your design, you'll need to save each of your documents and rename them accordingly. To save a document, right-click on the document's icon, then select **Save**



Rename each of the documents until they match the names below:



Congratulations! You just finished creating your first Project! Now, you can get started on the schematic by accessing the tutorial: [2. Creating a Schematic](#)

Revision #12

Created 2026-10-07 17:22:31 UTC by mattgonzales

Updated 2026-10-07 21:55:05 UTC by mattgonzales