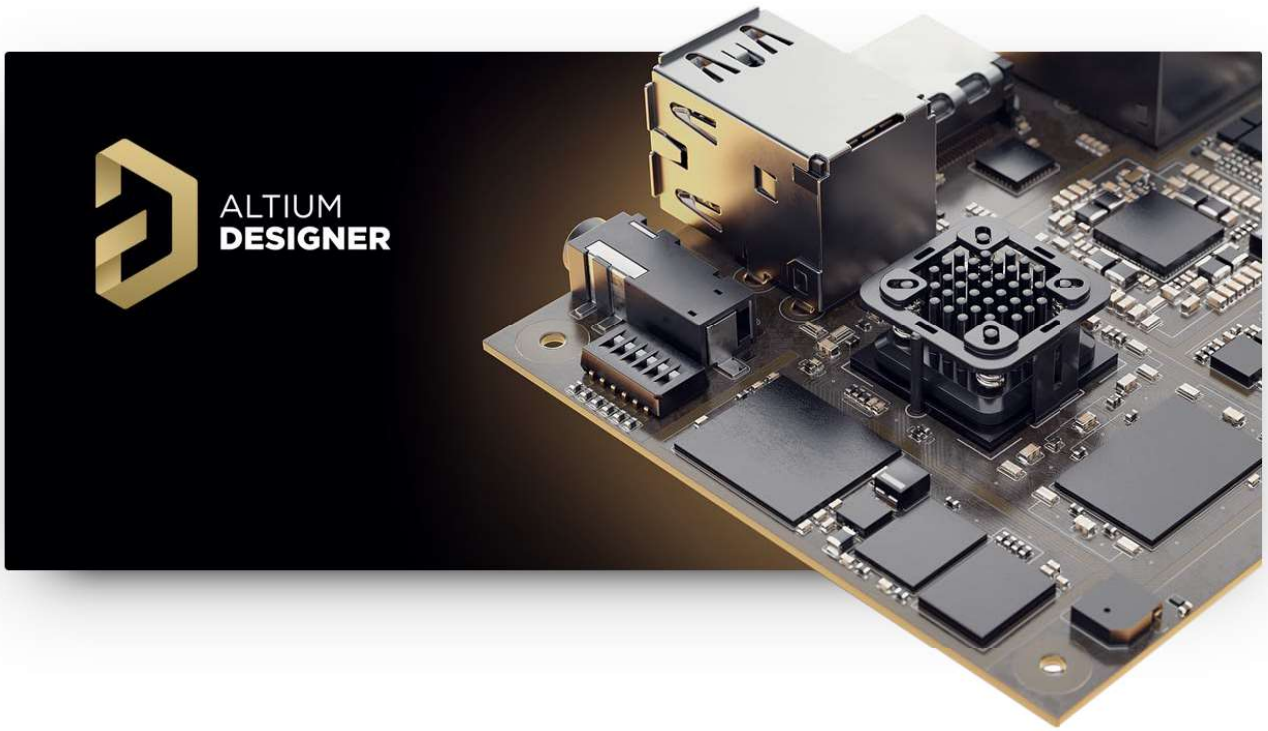

Altium Designer 20

Essentials Course

Table of Contents

Module 1: Design Environment

1.1 Purpose

In this exercise, you will learn to use some general GUI and navigation functions. You will have mastered the basic operations of the Altium Designer Environment after the completion of this exercise. This will be the foundation for future studies.

Altium Designer is a unified design environment. When you run Altium Designer for the first time, it will become readily apparent through your first steps of working in the software that all EDA tools are included in a single application. This includes schematic design, simulation, signal integrity analysis, PCB design tools, documentation/fabrication/assembly outputs, and library creation.

1.2 Opening Files

1. Click **File » Open Project.** Select **Local Projects** and browse to the example projects folder, which is installed by default in `C:\Users\Public\Public Documents\Altium\<AD Version>\Examples\ Bluetooth Sentinel.`
2. Open the `Bluetooth_Sentinel.PrjPcb` from its Bluetooth Sentinel sub-folder as shown in Figure 1 below.

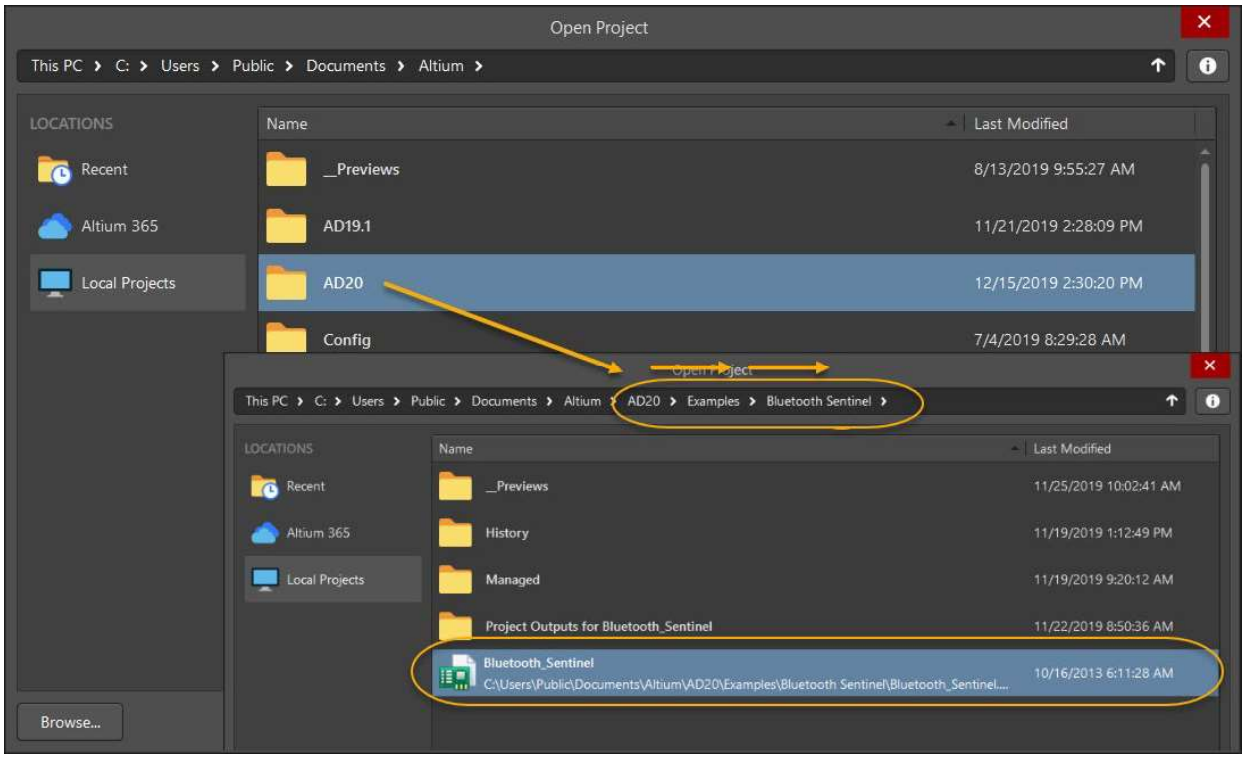

Figure 1. Open Project Dialog

NOTE: If this is your first time opening Altium Designer, some panels may appear in the middle of your screen. They can be closed by clicking the **"x"** in the top-right corner of the panel, or by right-clicking the title bar of the panel, and **Close.**

3. Locate the *Projects* panel on the left of the window as shown in Figure 2 below and double-click on the schematic file `Microcontroller_STM32F101.SchDoc`. This will open the schematic document within the active workspace and provide the working view of the schematic.
4. In the Projects panel, double-click on the PCB file `Bluetooth_Sentinel.PcbDoc`. This will open the PCB document within the active workspace and provide the working view of the PCB.

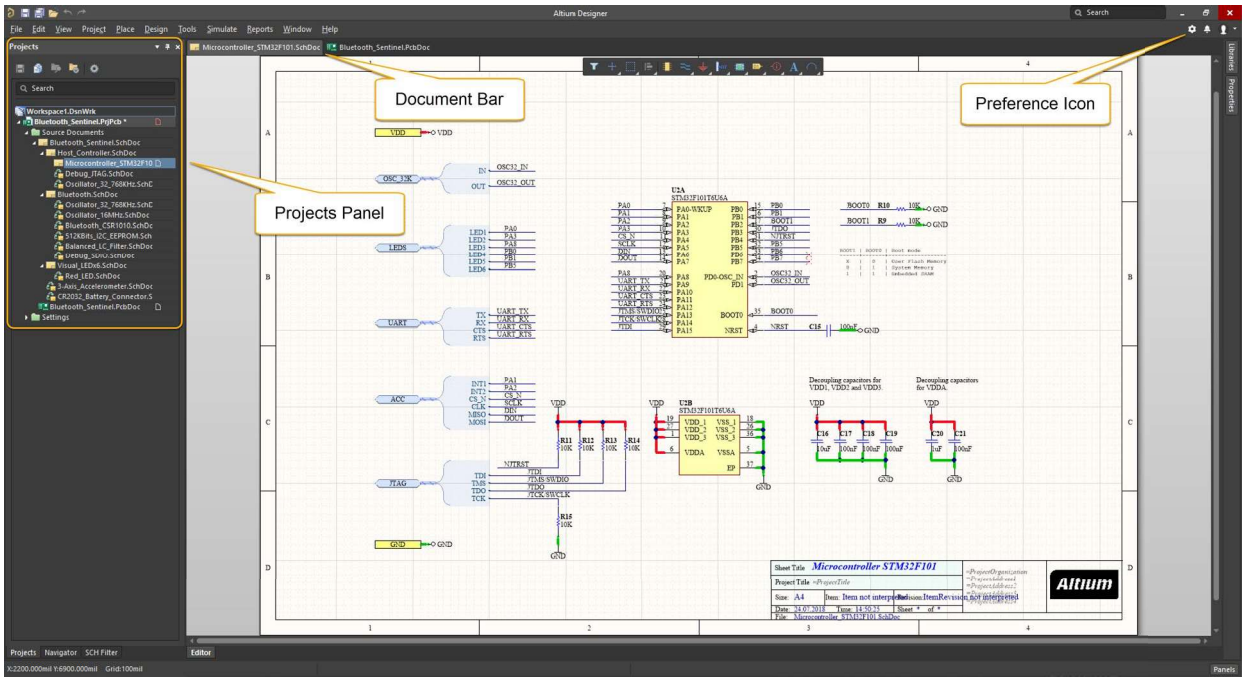

Figure 2. Altium Designer Environment

5. Notice that there are now two tabs on the Document Bar at the top of the editor. Click on each of these tabs and observe the workspace change to match the document that you are focused on.
6. Hit the shortcut keys **Ctrl+Tab** or **Ctrl+Shift+Tab** to cycle between the documents.
7. If we open multiple documents to the point where the tabs fill up the entire bar, we can configure its grouping method. Click the *Preferences* icon on the upper right corner of Altium Designer. Expand the *System* branch and open the *View* page. Within the *Documents Bar* settings, you can enable the **Group documents if need** checkbox and set it to **By document kind** as shown in Figure 3.