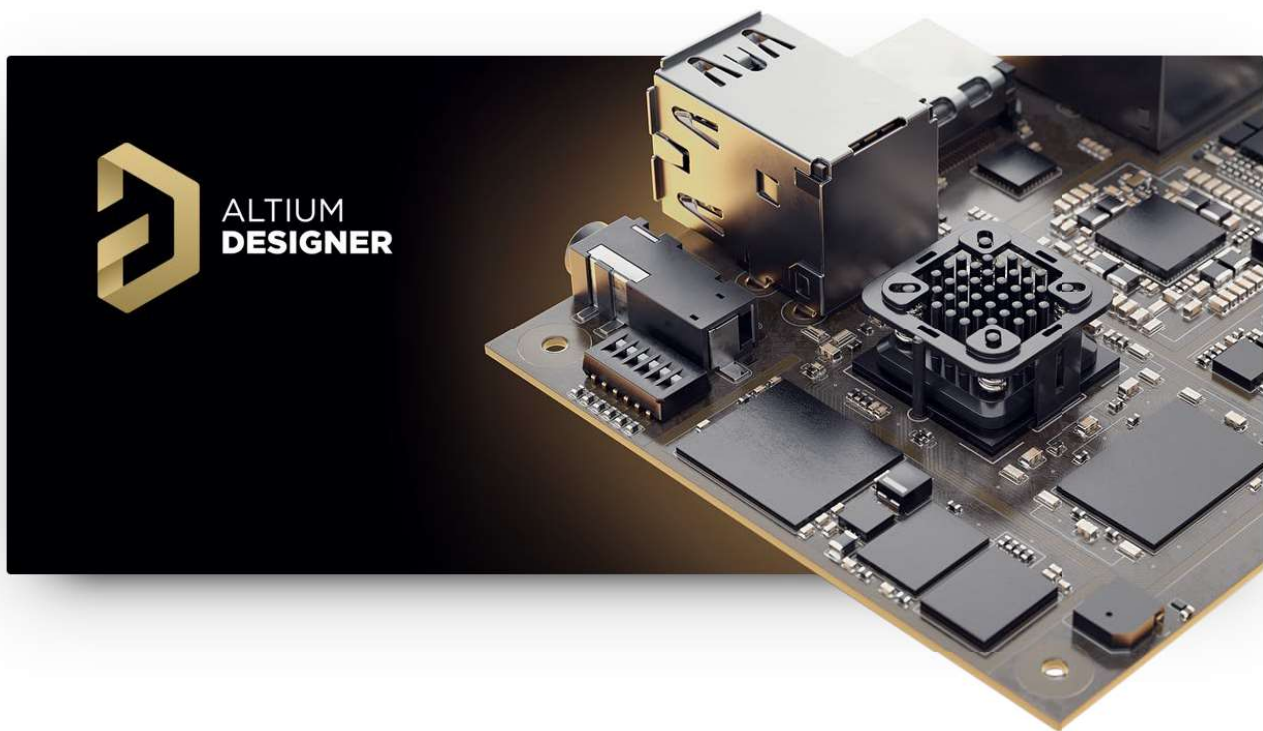




Altium Designer 20

Essentials Course

Software, documentation and related materials:

Copyright © 2020 Altium LLC

All rights reserved. You are permitted to use this document provided that (1) the use of such is for personal use only and will not be copied or posted on any network computer or broadcast in any media, and (2) no modifications of the document are made. Unauthorized duplication, in the whole or part, of this document by any means, mechanical or electronic, including translation into another language, except for brief excerpts in published reviews, is prohibited without the express written permission of Altium LLC. Unauthorized duplication of this work may also be prohibited by local statute. Violators may be subject to both criminal and civil penalties. Altium®, Altium Designer®, Altium Vault®, AutoTrax®, Camtastic®, CircuitMaker®, CircuitStudio®, Codemaker™, Common Parts Library™, DXP™, Easytrax®, LiveDesign®, NanoBoard®, Octopart®, PCBWORKS®, P-CAD®, Protel®, TASKING® and their respective logos are trademarks or registered trademarks of Altium LLC or its subsidiaries. All other registered or unregistered trademarks referenced herein are the property of their respective owners and no trademark rights to the same are claimed.

Table of Contents

Table of Contents	3
Module 1: Design Environment	14
1.1 Purpose	14
1.2 Opening Files	14
1.3 Documents & Panels	17
1.3.1 Document Bar	17
1.3.2 Panel Access	18
1.3.3 Panel Display Modes	18
1.3.3.1 Docked Mode	18
1.3.3.2 Pop-up Mode	19
1.3.3.3 Floating Mode	20
1.3.4 Grouping Panels	21
1.3.4.1 Tabbed Panel Grouping	21
1.3.4.2 Fractal Panel Grouping.....	22
1.3.5 Closing Panels	22
1.3.6 Maximizing/Restore Panels	22
1.3.7 Desktop Layouts.....	23
1.4 Toolbars	23
1.4.1 Active Bar.....	23
1.4.2 Classic Toolbars	24
1.5 Menu Operations	25
1.5.1 Right-Click Context Menu	25
1.5.2 Shortcut Keys	25
1.6 Status Bar and Command Status	26
1.7 Undo and Redo	26
Module 2: It Starts with a Project	29
2.1 Purpose	29
2.2 Create a New Project	29
2.2.1 Project Wizard.....	29
2.2.2 Adding New Documents	30
2.2.3 Project Options	32
2.3 Package a Project for Easy Transfer	33

Module 3: Schematic Preferences	34
3.1 Purpose	34
3.2 Preparation	34
3.3 Setting Preferences	34
3.4 Graphical Editing	39
3.4.1 Always Drag	39
3.4.2 Convert Special Strings	39
3.5 Grids	42
3.6 Schematic Defaults	43
3.6.1 Default Primitives	43
Module 4: Navigating Schematics	45
4.1 Purpose	45
4.2 Preparation	45
4.3 Basic Schematic Preference Settings	45
4.3.1 Setting the Default Template & Sheet Size	45
4.3.2 Project Insight.....	47
4.3.3 Connectivity Insight	49
4.3.4 Schematic Navigator	53
4.4 Navigation within the Schematic Workspace	54
4.4.1 Basic Navigation using the Mouse.....	54
4.4.2 Basic Navigation using the Keyboard	54
4.4.3 View Menu Hotkey Sequences	55
4.4.4 Jump Menu Hotkey Sequences	55
4.4.5 Search with the Search Field.....	56
Module 5: Schematic Capture	57
5.1 Purpose	57
5.2 Preparation	57
5.3 Create A New Project	57
5.3.1 Adding New Documents	58
5.3.2 Adding an Integrated Library.....	59
5.4 Component Placement	62

5.4.1 Setting the grid	62
5.4.2 Processor Interface Schematic Component Placement.....	62
5.4.3 CAN_Interface Schematic Component Placement	65
5.4.4 Relay_IO Schematic Component Placement.....	69
5.5 Adding an Existing Schematic to Your Design	71
Module 6: Making the Connection	72
6.1 Purpose	72
6.2 Preparation	72
6.3 Wiring the Schematic	72
6.3.1 Shortcuts	72
6.3.2 Processor_Interface Schematic	72
6.3.2.1 Wire Connections	72
6.3.2.2 Smart Paste Wires.....	74
6.3.2.3 Power Ports	74
6.3.2.4 Net Labels	75
6.3.2.5 Ports	76
6.3.3 CAN_Interface Schematic	78
6.3.3.1 Net Labels	78
6.3.3.2 Smart Paste Ports	78
6.3.4 Digital_IO Schematic	79
6.3.4.1 Auto-Generated Wires.....	79
Module 7: Schematic Updating	81
7.1 Purpose	81
7.2 Align	81
7.2.1 Align Right	81
7.3 Formatting Net Labels	81
7.3.1 Selecting All Net Labels	81
7.3.2 Apply Formatting to Net Labels	82
7.4 Locking Components Designators	83
7.4.1 Selecting only 100nF 50V Capacitors	83
7.4.2 Lock the 100nF 50V Capacitors Designators	84
7.5 Net Color Highlighting	85
7.6 Schematic Annotation	87
Module 8: Schematic Graphics	89

8.1 Purpose	89
8.2 Preparation	89
8.3 CAN Interface Schematic	89
8.3.1 Text String	89
8.3.2 Round Rectangle	90
8.4 Number Schematic Sheets	94
8.5 Smart PDF	96
Module 9: Compiling in Altium Designer	98
9.1 Purpose	98
9.2 Preparation	98
9.3 Project Options	98
9.3.1 Error Reporting Tab.....	98
9.3.2 Connection Matrix Tab	100
9.3.3 Options Tab.....	101
9.4 Running the Compiler	102
9.4.1 Correcting Errors and Warnings	102
9.5 Dynamic Compiler	106
9.5.1 Enabling the Dynamic Compiler.....	106
Module 10: Adding Hierarchy	109
10.1 Purpose	109
10.2 Preparation	109
10.3 Create Top-Level Schematic	109
10.4 Sheet Symbols	109
10.4.1 Creating a Sheet Symbol from Scratch	109
10.4.2 Adding Sheet Entries	111
10.4.2.1 Use the Smart Paste to convert Ports to Sheet Entries	111
10.4.3 Creating a Sheet Symbol from Existing Sheet.....	113
10.5 Connecting the Sheet Symbols	114
10.5.1 Wiring the Sheet Symbols	114
10.5.2 Wiring the Relay Bus.....	114
10.5.3 Labeling the Bus and Bus Slices	114

10.6 Creating a Schematic Sheet from Sheet Symbol	115
10.6.1 Create Sheet from Sheet Symbol	115
10.6.2 Completing the Newly Created Schematic Sheet	115
Module 11: Transfer to PCB	118
11.1 Purpose	118
11.2 Preparation	118
11.3 Before the Transfer	118
11.3.1 Footprint Validation	118
11.3.1.1 Connect to the Content Vault	119
11.3.1.2 Install the WCT.IntLib library	120
11.4 Transferring the Design	121
11.4.1 The Initial Transfer.....	121
11.4.2 Cross Select Mode.....	121
11.4.3 Assigning Directives	122
Module 12: Updating the PCB	126
12.1 Purpose	126
12.2 Preparation	126
12.3 Component Links	126
Module 13: PCB Introduction	128
13.1 Purpose	128
13.2 Preparation	128
13.3 Moving PCB View	128
13.4 View Commands	129
13.5 General	129
13.6 DRC Violations Display	132
13.7 Interactive Routing	133
Module 14: PCB Viewing and Navigation	135
14.1 Purpose	135
14.2 Preparation	135
14.3 Jump Command	135

14.4 Cross Probe Mode	136
14.5 Cross Select Mode	137
14.6 3D Layout Mode	138
14.7 PCB Layers	139
14.8 View Configurations	140
14.9 Board Insight Display	142
14.10 Display	145
14.11 Board Insight Mode	146
Module 15: PCB Panels and Global Editing	147
15.1 Purpose	147
15.2 Preparation	147
15.3 Properties Panel	147
15.3.1 Modify PCB Traces	147
15.4 Filter Panel	149
15.4.1 Object Checkboxes	150
15.4.2 Query Selection	151
15.5 Find Similar Objects	154
15.5.1 PCB Find Similar Objects Dialog	154
15.5.1.1 Selecting Vias	154
15.5.1.2 Changing Via Size	155
15.6 List Panel	157
15.6.1 Selecting Components	157
15.6.2 Modifying Properties	158
Module 16: PCB Grids	160
16.1 Purpose	160
16.2 Preparation	160
16.3 Predefined Default Grids	160
16.4 Grid Manager	161
16.4.1 Global Board Snap Grid	161
16.4.2 Custom Cartesian Grid.....	163

16.4.3 Custom Polar Grid	166
Module 17: PCB Layers	168
17.1 Purpose	168
17.2 Preparation	168
17.3 Adding a new PCB	168
17.4 Layer Stack Manager	169
17.4.1 Modify Default Layer Stack	169
17.5 Layer Stack Table	172
Module 18: Shaping the PCB	174
18.1 Purpose	174
18.2 Preparation	174
18.3 Board Planning Mode	174
18.4 Define Board Shape from Selected Objects	178
18.4.1 Circle	178
18.4.2 Rectangle	180
18.5 Defining Board Shape from DXF/DWG	183
Module 19: PCB Design Rules	186
19.1 Purpose	186
19.2 Preparation	186
19.3 Basic Design Rule	186
19.3.1 PCB Rules and Constraints Editor	186
19.3.2 Rule Priorities	187
19.4 Design Rule Using Net Classes	189
19.5 Design Rule Using Custom Queries	192
19.5.1 Create Queries Using Find Similar Objects	192
19.5.2 Verifying Applied Design Rules in the PCB	194
19.6 Design Rule Wizard	196
Module 20: PCB Placement	198
20.1 Purpose	198

20.2 Preparation	198
20.3 Shortcuts	198
20.4 Placement of Components Within Rooms	199
20.4.1 Placing within an Area	200
20.4.2 Placing Individual Components	202
20.4.3 Cross Select Mode.....	203
20.4.4 Position Remaining Components	206
Module 21: PCB Routing	208
21.1 Purpose	208
21.2 Preparation	208
21.3 Shortcuts	208
21.4 PCB Routing	209
21.4.1 Setting Preferences	209
21.4.2 Quick Routing for Simple Nets	209
21.4.3 Interactive Routing Nets	209
21.4.4 Vias	211
21.4.4.1 Setting Via Sizes	211
21.4.4.2 Adding Vias	211
21.4.5 Multi-Trace Routing	212
21.4.6 Adjusting Tracks.....	214
21.4.7 Contour Follow Mode	214
21.4.8 Track Glossing.....	214
21.4.9 Active Route.....	215
Module 22: Polygon Pours	218
22.1 Purpose	218
22.2 Preparation	218
22.3 Placing a Polygon Pour	218
22.4 Polygon Properties	220
22.5 Polygon Pour Manager	222
22.6 Flooding a Layer	224
22.7 Editing Polygons	226
22.7.1 Editing an Edge	226

22.7.2 Adding a Vertex	227
22.7.3 Deleting a Vertex.....	227
22.8 Controlling the Connection Style - Polygon	228
22.9 Power Planes	230
22.9.1 Controlling the Connection Style – Power Plane.....	231
Module 23: DRCs on PCBs	232
23.1 Purpose	232
23.2 Preparation	232
23.3 Design Rule Check	232
23.4 PCB Rules and Violations Panel	234
23.5 Resolving Violations	236
Module 24: Draftsman	240
24.1 Purpose	240
24.2 Preparation	240
24.3 Creating a Draftsman Document	240
24.3.1 Creating Draftsman From a Template.....	240
24.3.2 Creating Draftsman Without a Template	241
24.4 Page Configuration	241
24.5 Assembly View	243
24.6 Fabrication View	245
24.7 Additional Views and Dimensions	246
24.8 Tables	248
24.8.1 Optional	248
Module 25: PCB Fabrication Files	249
25.1 Purpose	249
25.2 Preparation	249
25.3 Bill of Materials	249
25.4 Output Job File	251
25.4.1 Creating a New Output Job File	251

25.4.2 Defining Outputs.....	251
25.4.2.1 Documentation Outputs.....	251
25.4.2.2 Assembly Outputs	252
25.4.2.3 Fabrication Outputs	253
25.4.2.4 Report Outputs	253
25.4.2.5 Validation Outputs	253
25.4.3 Defining Containers.....	254
25.4.3.1 PDF Container.....	254
25.4.3.2 Folder Container	254
25.4.4 Generating the Outputs.....	255
25.4.4.1 Generating the PDF.....	255
25.4.4.2 Generating the Folder Structure.....	255
<u>Module 26: Schematic Library</u>	<u>257</u>
<u>26.1 Purpose</u>	<u>257</u>
<u>26.2 Create a Crystal Symbol</u>	<u>257</u>
26.2.1 Creating a Schematic Library	257
26.2.2 Symbol Properties	257
26.2.3 Defining the Pins	259
26.2.4 Adding Symbol Body	261
<u>26.3 Using the Symbol</u>	<u>263</u>
26.3.1 Installing the Library	263
26.3.2 Adding Symbol to Schematic	264
<u>26.4 Revising the Symbol</u>	<u>265</u>
26.4.1 Making a Change to the Symbol	265
26.4.2 Updating the Schematic.....	265
<u>Module 27: PCB Library</u>	<u>266</u>
<u>27.1 Purpose</u>	<u>266</u>
<u>27.2 Creating a PCB Library</u>	<u>266</u>
<u>27.3 Footprint Creation</u>	<u>267</u>
27.3.1 Setting the Grid.....	268
27.3.2 Adding Pads.....	269
27.3.3 Setting Component Origin	270
27.3.4 Adding Top Overlay Outline	271
27.3.5 Add Pin 1 Marker	272
27.3.6 3D Crystal Body	273
<u>27.4 Use the Footprint</u>	<u>275</u>

27.4.1 Installing the Footprint Library	275
27.4.2 Adding Crystal Footprint to PCB Document	276
Module 28: Component Creation	277
28.1 Purpose	277
28.2 Linking Footprint to the Symbol	277
28.3 Using Linked Component	279
Module 29: Integrated Library	282
29.1 Purpose	282
29.2 Preparation	282
29.3 Create Integrated Library Project	282
29.4 Using the New Library	286

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.

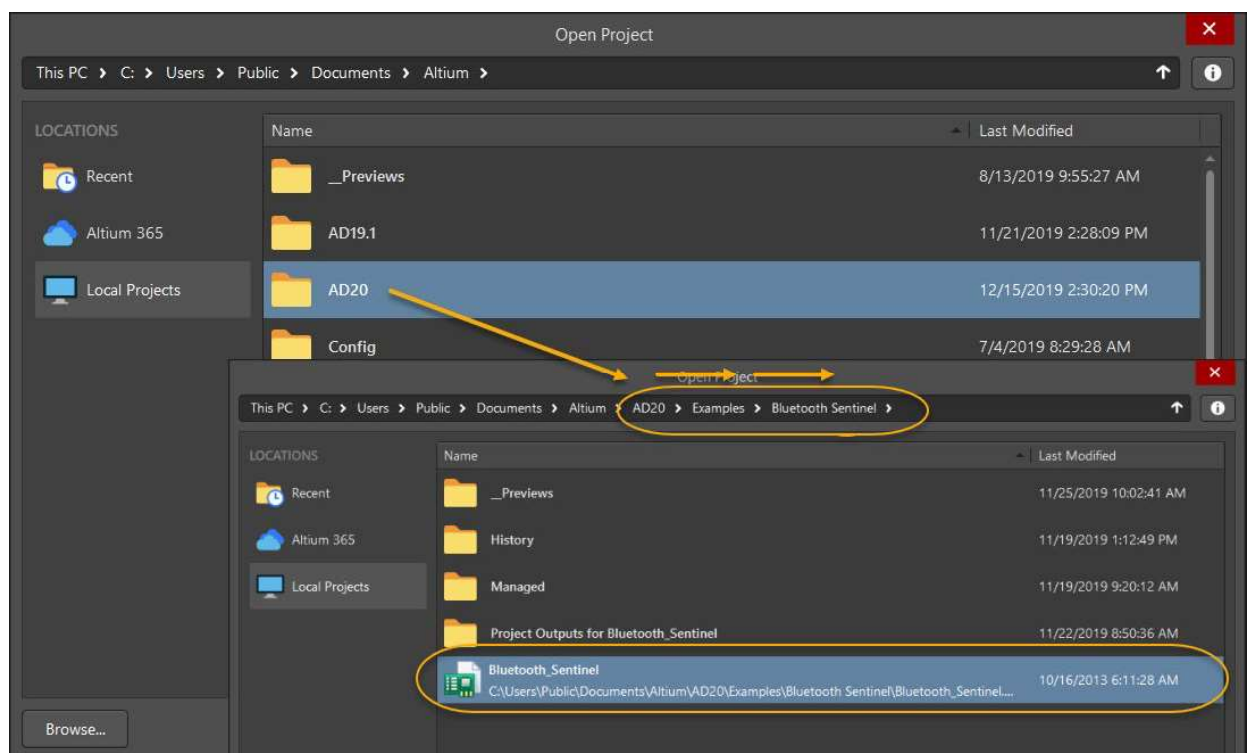


Figure 1. Open Project Dialog



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.

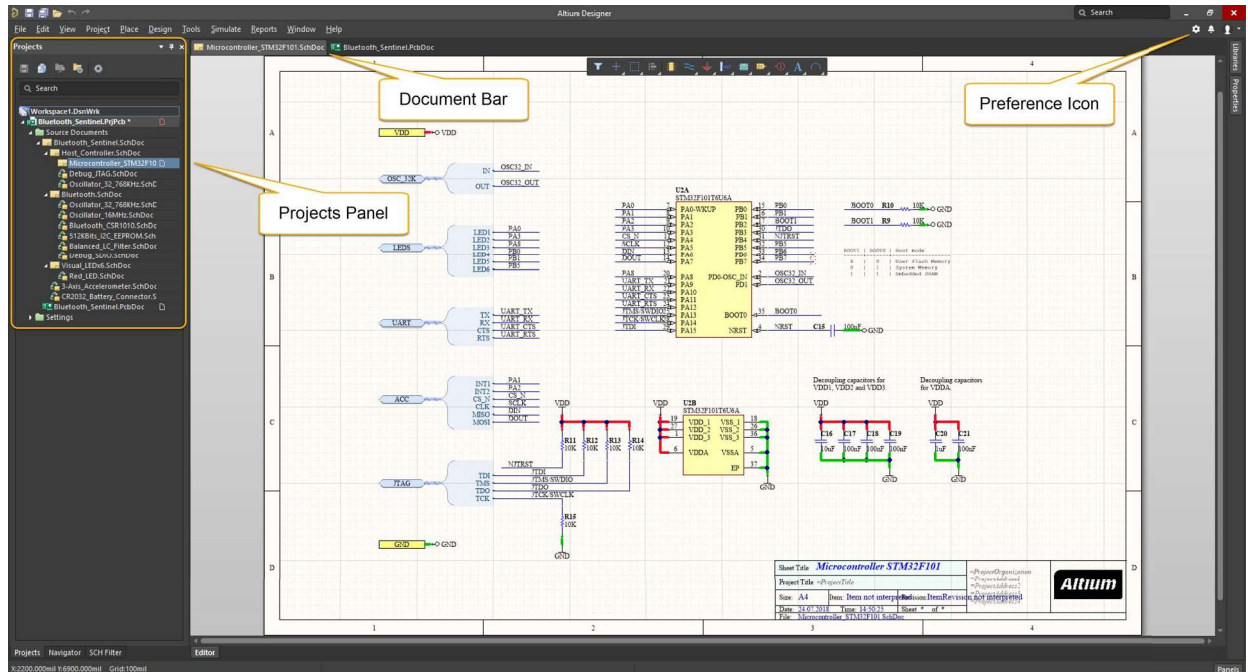


Figure 2. Altium Designer Environment

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.