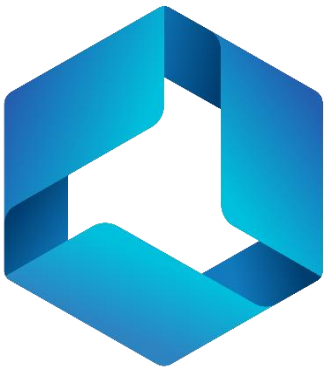

ALTIUM 365

Altium Designer

Essentials Course - Altium 365

Module 15: PCB Introduction

Table of Contents

Module 15: PCB Introduction

1.1 Purpose

In this exercise, you'll explore some of the PCB specific general preference settings and learn to customize them. You will also briefly use view commands, such as navigating to specific objects and more.

1.2 Shortcuts

Shortcuts when working with Module 15: PCB Introduction

1:	Board planning mode
2:	2D mode
3:	3D mode
8:	3D Orthogonal Rotation view.
9:	3D 90 Degree Rotation
0:	3D Zero Rotation
Shift+right click:	3D Navigation
or **T » P:**	Preferences
ALT key and left-click:	Highlight Net
Shift+C:	Clear Masking and/or Selection
V » D:	View » Fit Document
V » F:	View » Fit Board
J » C:	Jump to Component
J » L:	Jump to Lcation
Hovering:	Show Object information (Head Up Display)
Mouse Wheel:	Scroll up - Scroll down
Mouse Wheel+Ctrl:	Scroll left - Scroll right
Mouse Wheel+Shift:	Zoom in - Zoom Out
Mouse Wheel+Shift+Ctrl:	Layer change
" + " , " - " , or " * "	Layer Change (on the numerical keypad)
Press Mouse Wheel+ Mouse Move:	Zoom in - Zoom Out
Page Up:	Zoom in
Page Down:	Zoom out

1.3 Preparation

1. **Close all existing projects and documents.**
2. Next, create a Copy / Clone of the Training Project `Module 15 PCB Introduction.`
3. Select **File » Open Project…** to open the *Open Project* dialog.
4. Navigate to the predefined Training Project `Module 15 PCB Introduction (Top\Projects\Altium Designer Essentials Training Course\...).`
5. Select **Open Project as Copy…** 

.
6. At the new dialog *Create Project Copy:*
   a) Add your name to the project: `Module 15 PCB Introduction – [Your Name].`
   b) Add a description: `Altium Essential Training - Module 15 PCB – [Your Name].`
   c) Open the *Advanced* section.
   d) Select the Ellipsis Button 

from the **Folder** configuration to open the *Choose Folder* Dialog.
      i) Select the folder with your name: `Project\For Attendees\[Your Name]`
      ii) Select **OK**
   e) Change the Local Storage path if needed.
   f) Select **OK** to create the copy.
7. Wait until Altium Designer created the copy of the project and opened the project for you at the *Projects* panel, this may take up to 1 minute.

For details how to Copy / Clone the predefined training project see Module 8 Making the Connection, Step 1.3 Preparation.

1.4 Overview

8. Open at least one Schematic and the PCB.
9. A change from Sch to PCB or vice-versa is a change from one document type to another, this change of focus also changes the working environment and the menus available, as seen in Figure 1 and Figure 2

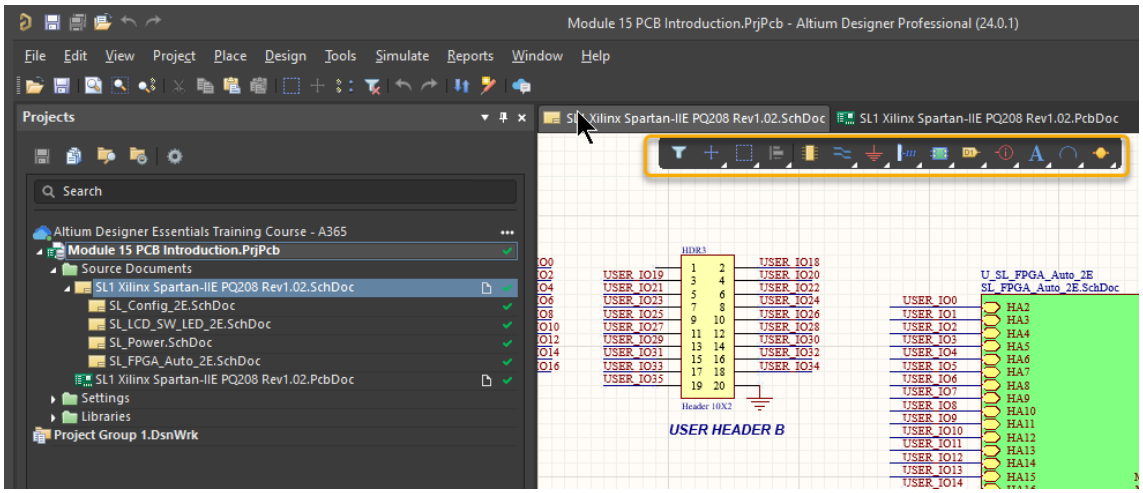

Figure 1. Schematic Design Environment

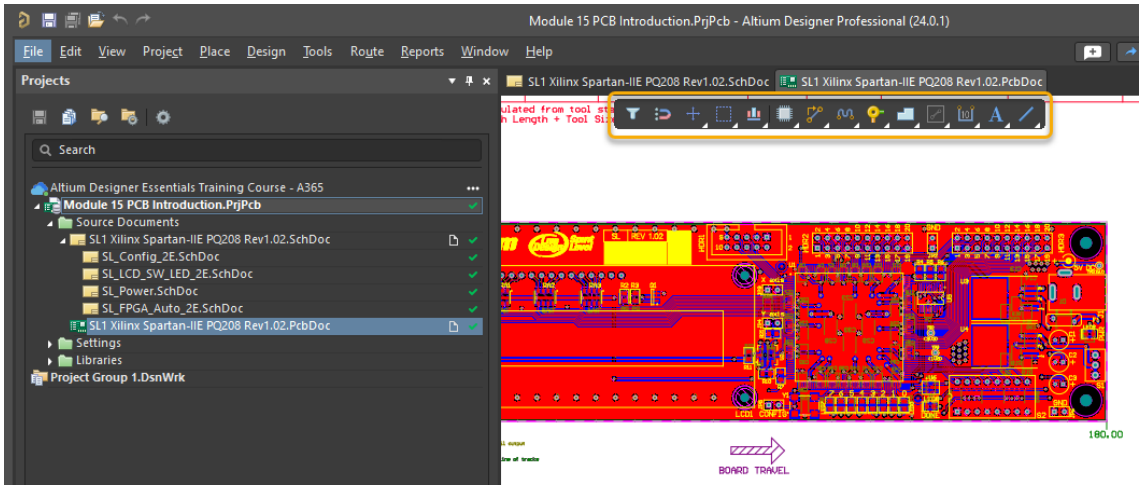

Figure 2. PCB Design Environment

1.5 PCB viewing Modes

1.5.1 **Board Planning Mode**

This Mode is primarily for Board Shape creation and modification in combination with Flex- Rigid Designs (for details about Flex Rigid join our Advanced Training).

10. Press **1** on the keyboard to switch to the Board Planning mode. You can find the command also in the View menu **View » Board Planning Mode,** Figure 3 left.

1.5.2 **2D Layout Mode**

This Mode is primarily for Component Placement, Routing, Polygon Pour Management, ….

11. Press **2** on the Keyboard to switch to the 2D Mode. You can find the command also in the View menu, **View » 2D Layout Mode,** Figure 3 center.

1.5.3 **3D Layout Mode**

This Mode is primarily the 3D View of the Board for Rigid and Flex-Rigid Boards. With 3D Bodies for the component, you can see the height profile of your PCB or for Flex-Rigid designs you can animate the fold.

12. Press **3** on the Keyboard to switch to the 3D Mode. You can find the command also in the View Menu, **View » 3D Layout Mode,** Figure 3 right.

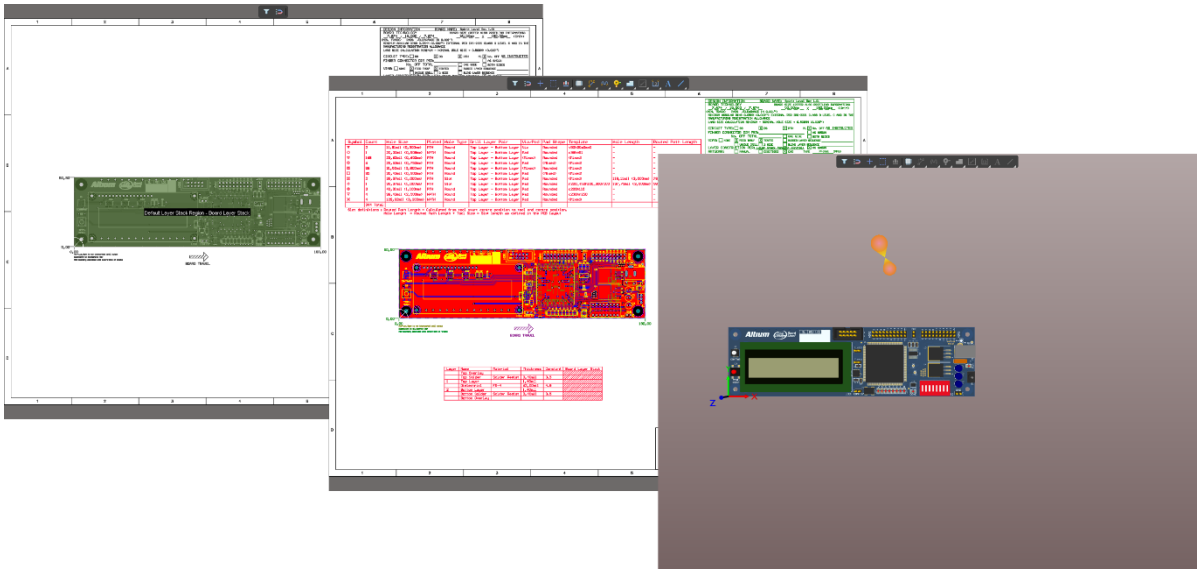

Figure 3. The three Viewing Modes, Board Planning Mode / 2D / 3D

1.6 Moving PCB View

The commands for moving around the PCB environment are very similar to the commands we use in the Schematic. The commands can be done in all three PCB View Modes.

13. To switch to the 2D workspace, where we do the majority of our work, hit the **2** key.
14. Hold the Right-Mouse-Button until the dragging hand cursor appears. Move your mouse while still holding the right mouse button to drag the PCB around, this is how we pan. Release the right button to end the drag operation.
15. Scroll the mouse-wheel forward or back to scroll the PCB vertically.
16. Hold **Shift** while scrolling the mouse wheel forward or back to scroll horizontally. Ignore the orb you see (in 3D Mode) if you press **Shift**, the functions in combination with the orb will be explained in paragraph **3D Layout Mode**.
17. Hold **Ctrl** while scrolling the mouse wheel forward or back to zoom in and out.
18. Press and hold the mouse wheel down and move your mouse to zoom in and out.

1.7 View Commands

The commands for viewing the PCB are very similar to the command we use in the SCH.

19. Maintain the focus on the PCB Workspace 2D Mode and observe the behavior of the following shortcut keys:
    a) **View » Fit Document** or press the **V » D** keys in succession.
    b) **View** » **Fit Board** or press the **V » F** keys in succession.
    c) **View » Zoom In** or press the **V » I** keys in succession or press **Page Up.**
    d) **View » Zoom Out** or press the **V » O** keys in succession or press **Page Down.**
    e) **View » Area** or press the **V » A** and left click to draw a rectangle in a diagonal manner.
    f) **View » Zoom Point** or press the **V » P** keys in succession, then left click to draw a rectangle in a diagonal manner. The contents of the rectangle will be focused and shown with the zoom level adjusted.

1.8 3D Layout Mode

21. While still being in the active PCB document, go to **View » 3D Layout Mode** or press the **3** key to enter 3D mode.

Numbers on the numerical keypad won't work for changing views presented with Step 21. Using the keys above the QWERTY keys will.

a) Press the **8** key to change the 3D view to "*Orthogonal Rotation*" view.
b) Press the **9** key or **0** key for "*90 Degree Rotation*" or "*Zero Rotation*".
c) Press and hold the **Shift** key until you see an orb as shown in Figure 4 below.

Figure 4. Orb to navigate in the 3D environment

d) While still holding **Shift,** press and hold the **right mouse button** on one of the navigation objects in the orb shown in Figure 5 below (arrow, circle, dot) and now move the mouse to change the 3D view.

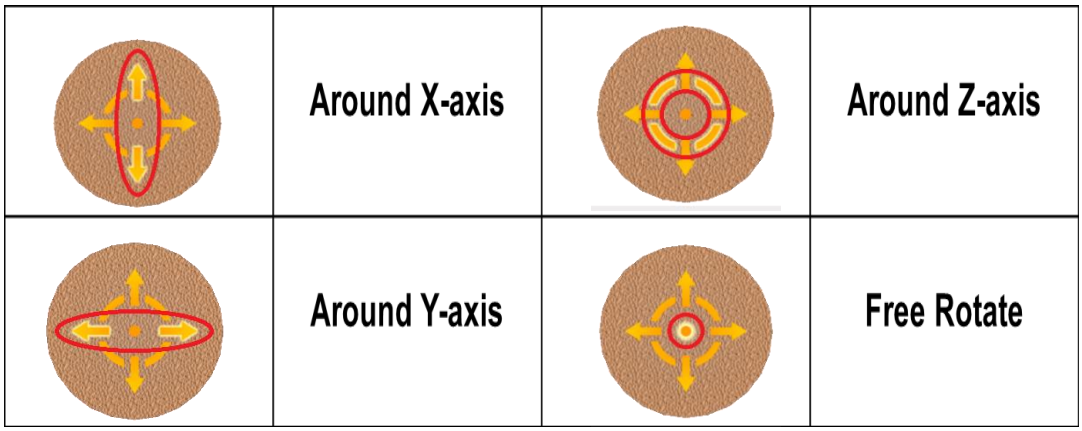

Figure 5. Orb controls for navigating in the 3D environment

22. Instead of using the orb you can control the 3D View with shortcut key / menu commands from the **View Menu**, see Figure 6.

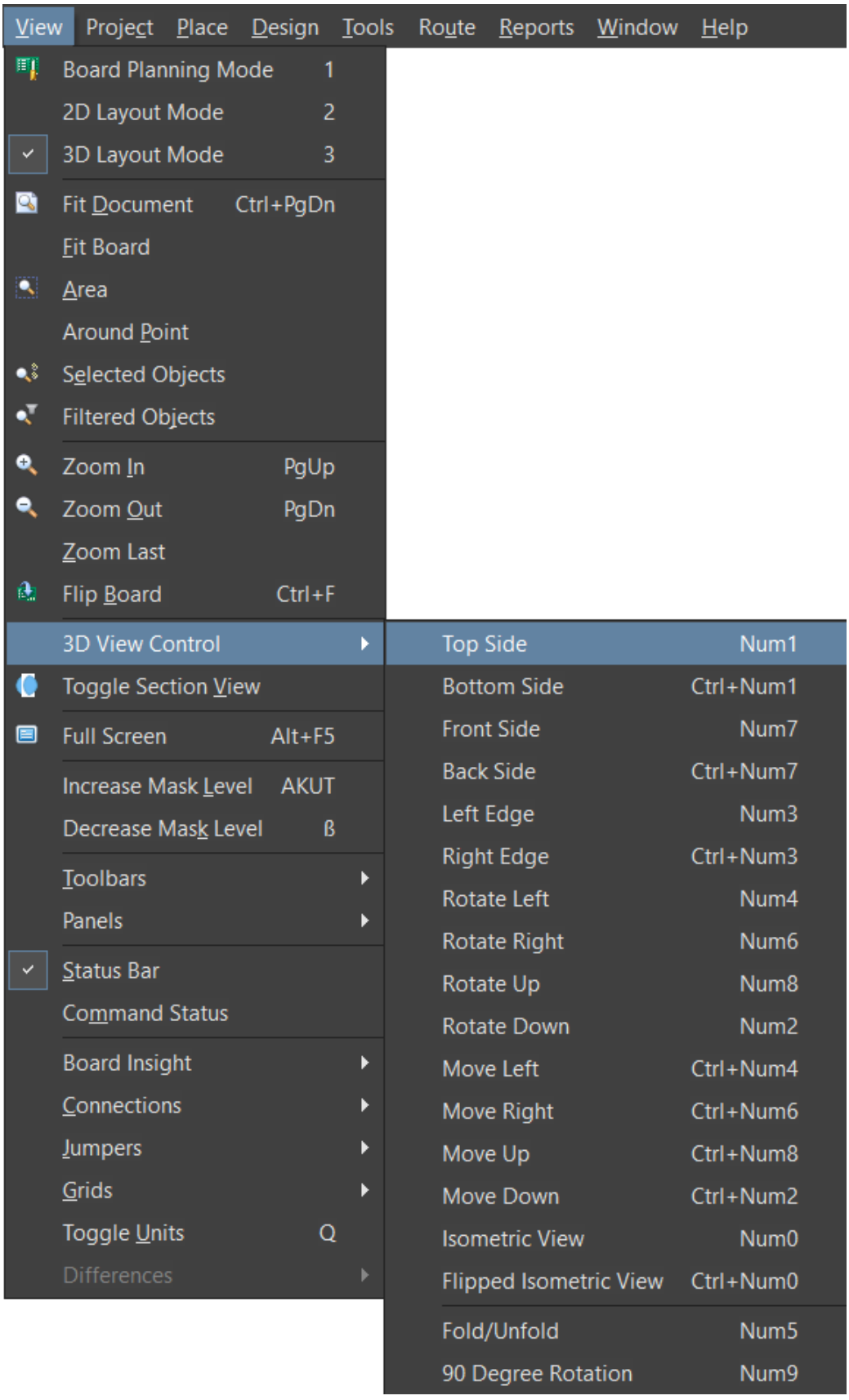

Figure 6. PCB 3D View Control

23. Select **View » 2D Layout Mode** or press the **2** key to go back to the 2D view mode.

1.9 PCB Layer

24. At the bottom of the PCB workspace, you find tabs that show the active layers from the PCB.
    a) Left-click to select the `Top Layer`. You will know the layer is the active layer once the layer text is in bold writing as shown in Figure 7**.** You can use the arrows to scroll through the available layers.
    b) Click on any other layer tab to change the active layer. See how the PCB view change based on your selection.
    c) Click on `Top Layer` tab again to activate the *Top Layer*.

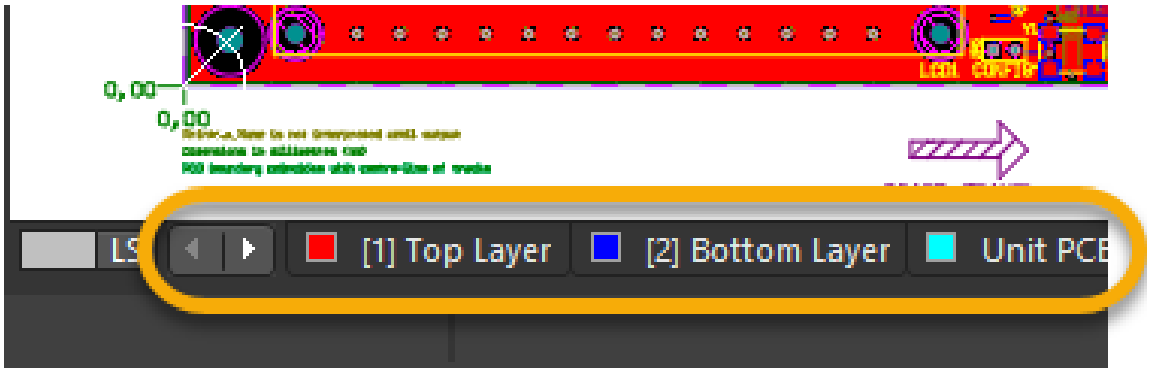

Figure 7. Ensure the Top Layer is Active

1.10 Preference Settings

1.10.1 General

25. Click the ⚙ in the upper right corner, to open the *Preferences*. Open the *PCB Editor* branch and open the *General* page as shown in Figure 8 below.

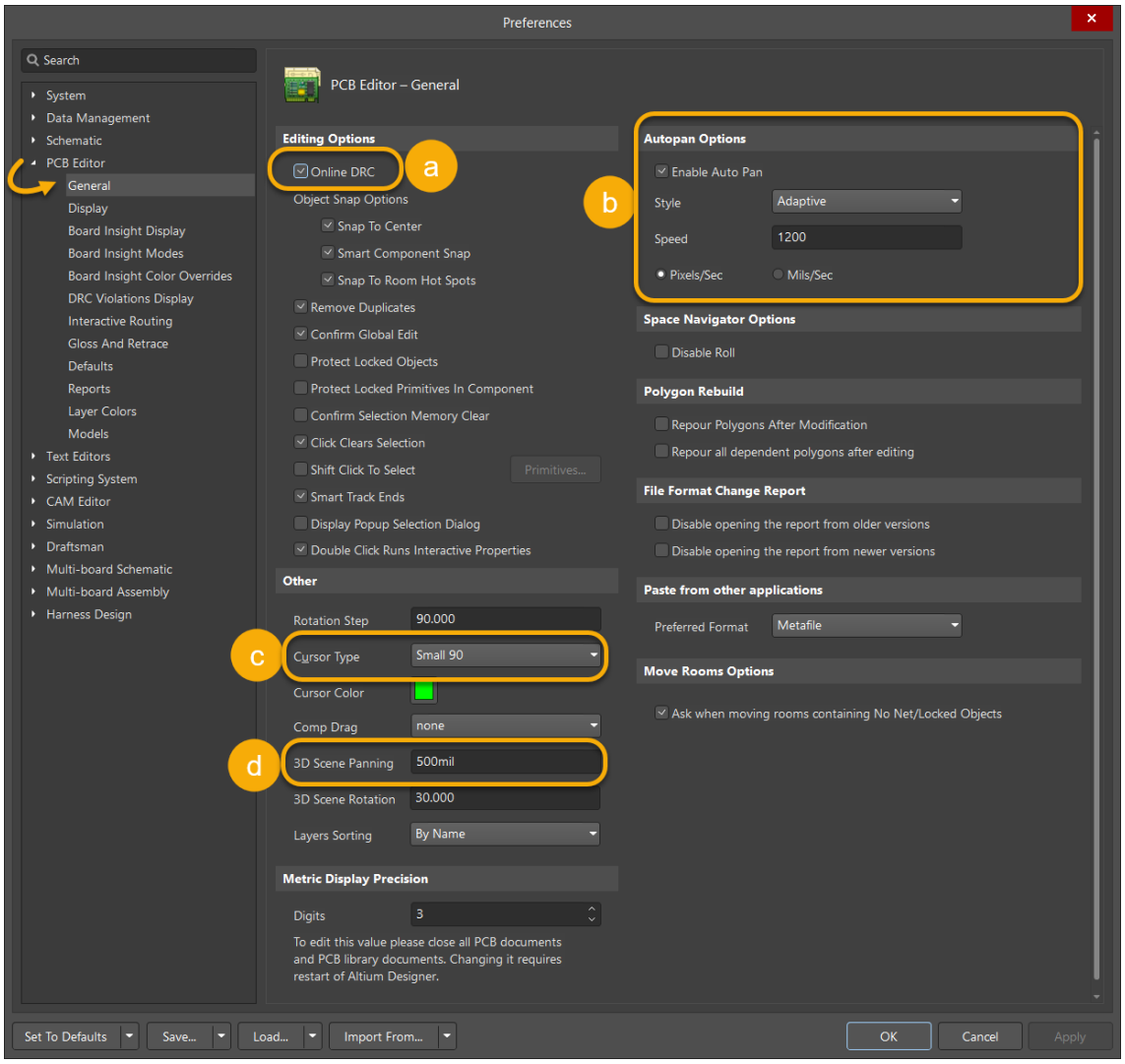

Figure 8. General Preferences for the PCB Editor

a) Verify that the **Online DRC** is enabled.

b) The *Autopan Options* allows you to control the usage and speed for panning actions. The panning style and speed depend on your personal preferences.

c) Change the *Cursor Type* to **Large 90.**

d) Configuration for the 3D View control done by numeric keypad shortcut commands.

1.10.2 **Board Insight Modes**

26. Change to the section *Board Insight Modes* in the *Preferences*, as seen in Figure 9.

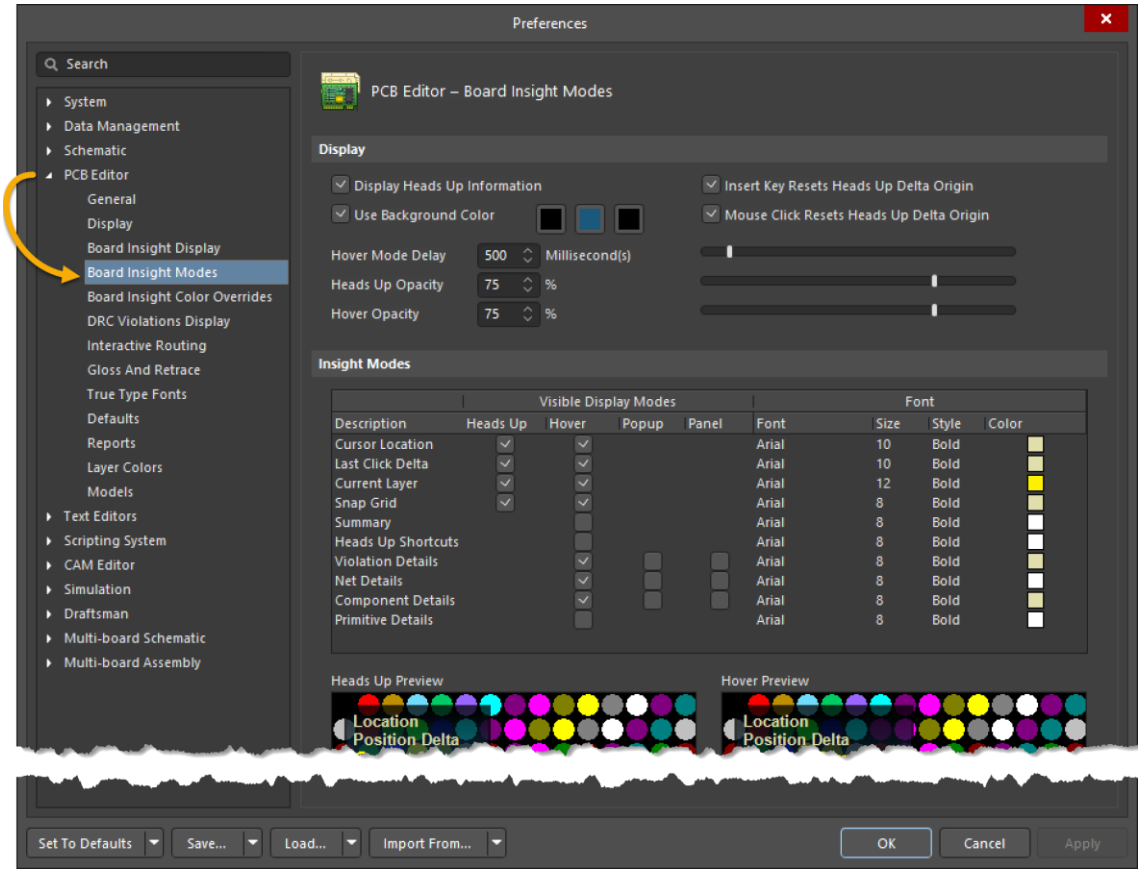

Figure 9. Board Insight Modes

27. The *Board Insight Modes* settings allows you to control and configure the information area in the PCB called *Heads Up Display*, see Figure 10.
28. Click **OK** to close the *Preferences* dialog and return to the PCB.
29. Hover over a component to see the *Heads-Up Display* with the component information, as shown in Figure 10.

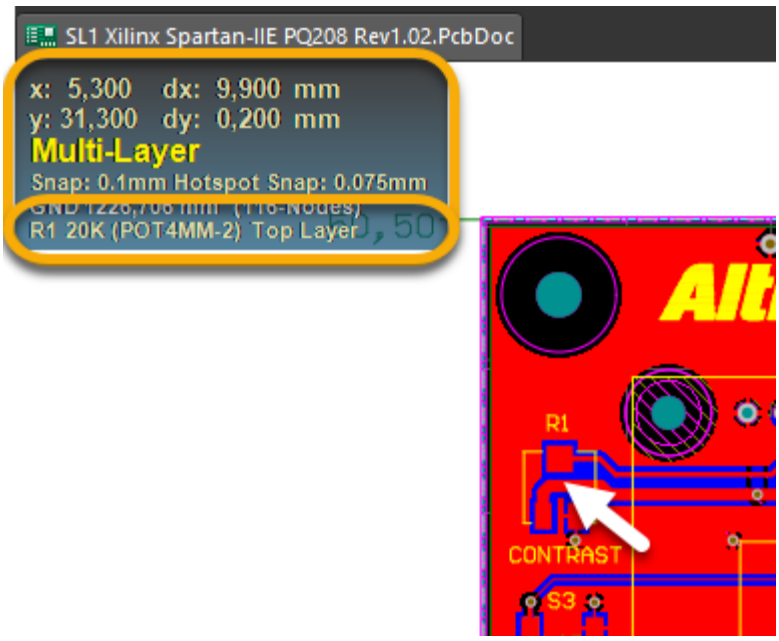

Figure 10. Heads Up Display

1.11 Properties Panel

The *Properties* panel allows you to see and configure the general PCB options, or the parameters of selected PCB objects.

30. Open the *Properties* panel, if not already open from the **Panels** button Panels, Figure 11.

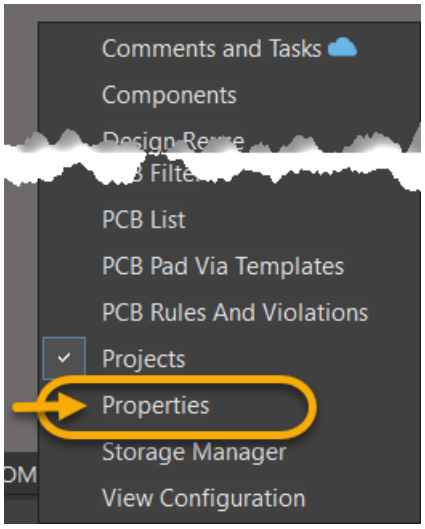

Figure 11. PCB panels Bottom to open Properties Panel

1.11.1 Selection Filter

31. The first pane allows you to control the selection filter for the PCB.
32. Enable the selection filter to only select **Components** and deactivate it for all other elements, as shown in Figure 12 below. If you do not see the selection filter (you see object parameters) click in a free area of the PCB to deselect everything or press **Shift+C.**

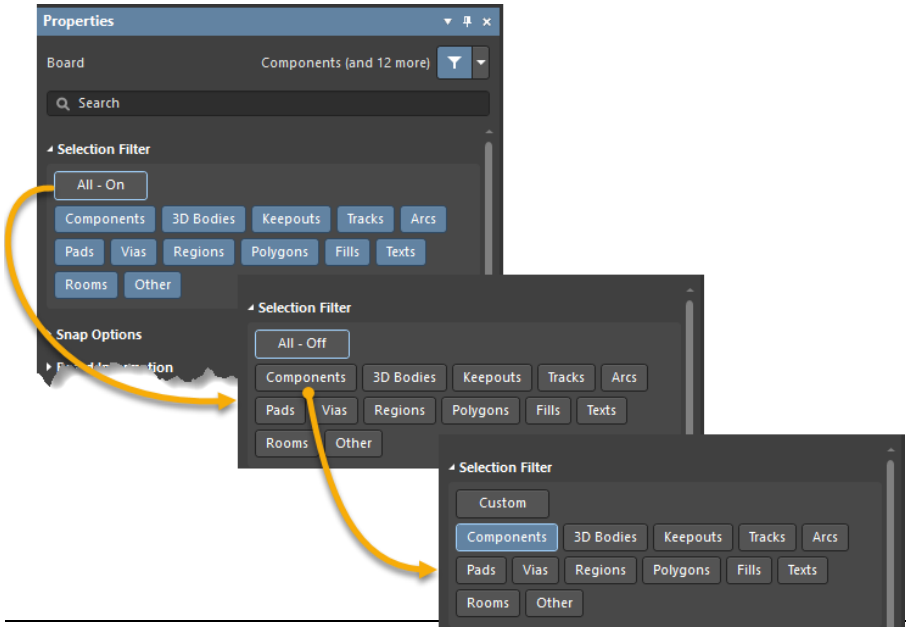

Figure 12. Selection Filter for Components

33. Draw a rectangle with your mouse to select objects in the PCB, with the *Selection Filter* set to **Components,** notice only components are now selected.
34. Reset the *Selection Filter* by clicking on **Custom**. *All-On* is shown to select all kind of objects again.
35. Click in a free area of the PCB or press **Shift+C** to deselect the currently selected objects.

1.11.2 **Board Information**

36. The Board information section gives you an overview over your PCB and the option to generate **Reports**, as seen in Figure 13.

1.11.3 **PCB Units**

37. You can change between metric and imperial Unit by clicking on **mm** or **mils** button in the P*roperties* panel, see Figure 13, or alternatively hit **Q** on the keyboard to change between the units in the active PCB document.

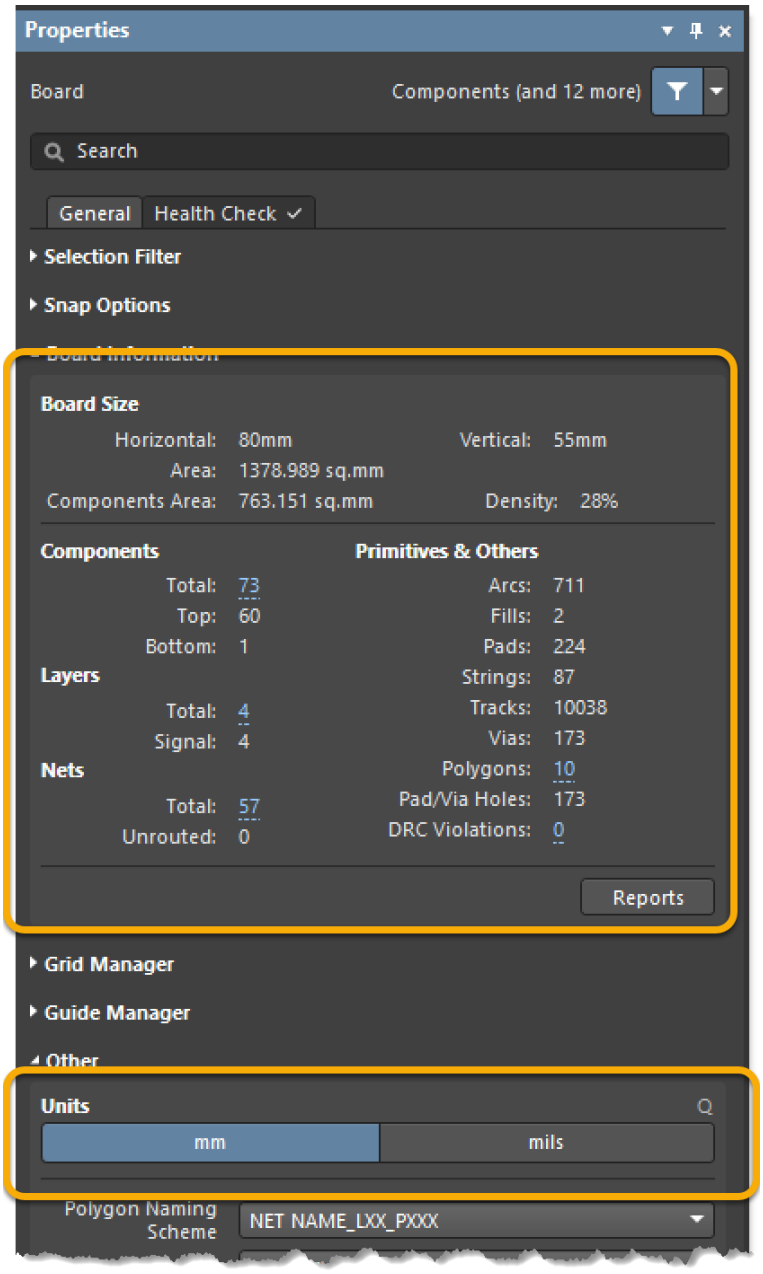

Figure 13. Properties Panel with Board Information and Units

38. **Do not close the project, we will use this project in the next module as well.**
39. Feel free to save the modifications you have done, using **File » Save All**.
40. Save the modifications to the server:
    a) At the *Project* panel, next to the Project name you find the command **Save to Server** 

    .
    b) Select **Save to Server.**

c) At the dialog *Save [Project Name],*
   i) Activate the checkboxes for the files that are not under version control.
   ii) Add the comment `Module 15: PCB Introduction - [Add Your Name] - Finished`.
   iii) Select **OK.**

Congratulations on completing the Module!

Module 15: PCB Introduction

from the

Altium Designer Essential Course
with Altium 365

Thank you for choosing Altium Designer