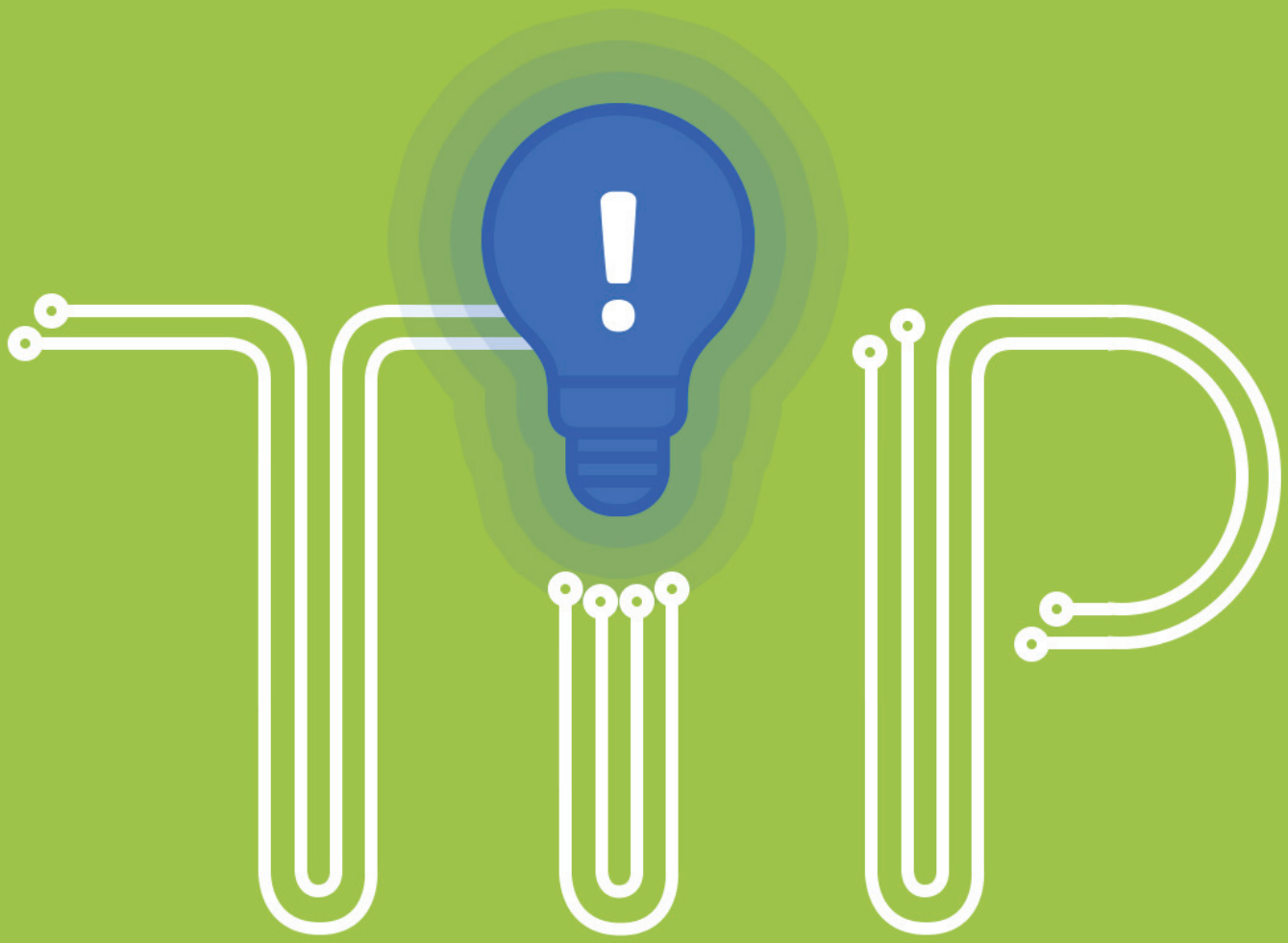
Altium

Group Components into Rooms for More Efficient Layout



Jesse Lai
Applications Engineer

GROUP COMPONENTS INTO ROOMS FOR MORE EFFICIENT LAYOUT

The key to properly managing your component placements and traces is to utilize different techniques of grouping objects together rather than individually modifying each individual object. A lot of users hate the idea of having to individually bring a component to the board layout. This paper details what Altium Designer® can do to make layout management easier and less time consuming, allowing you to meet your project deadlines.

INTRODUCTION

A component layout can become really messy if the components and traces are not organized properly. The most common methods to a managed design layout is through the use of rooms. Rooms can be used to manage component placements better, and it can easily help one identify the component's origin, which is explained more in detail below.

A rat's nest of connections can be a pain as well because if no routing was performed and a lot of components are used. It will consume additional resources to produce connection lines across the layout, significantly decreasing system performance and making component placement much more difficult.

USING ROOMS

Rooms are commonly used during design transfers from the schematic to the PCB editor, where each are defined as respective schematic sheets. The components are defined on each sheet as component classes, in which its generations are defined through the project's configurations. For example, a project contains 5 different sheets, each containing specific components, in which the importance of a flat versus hierarchical design is voided. When the schematic is pushed over to the PCB layout of the project, the PCB layout will contain the sheet-defined rooms with their respective components used, illustrated in Figure 1. For components that are not already placed within a room after ECO generation, you can manually define a room over them, or drag components to the new room.

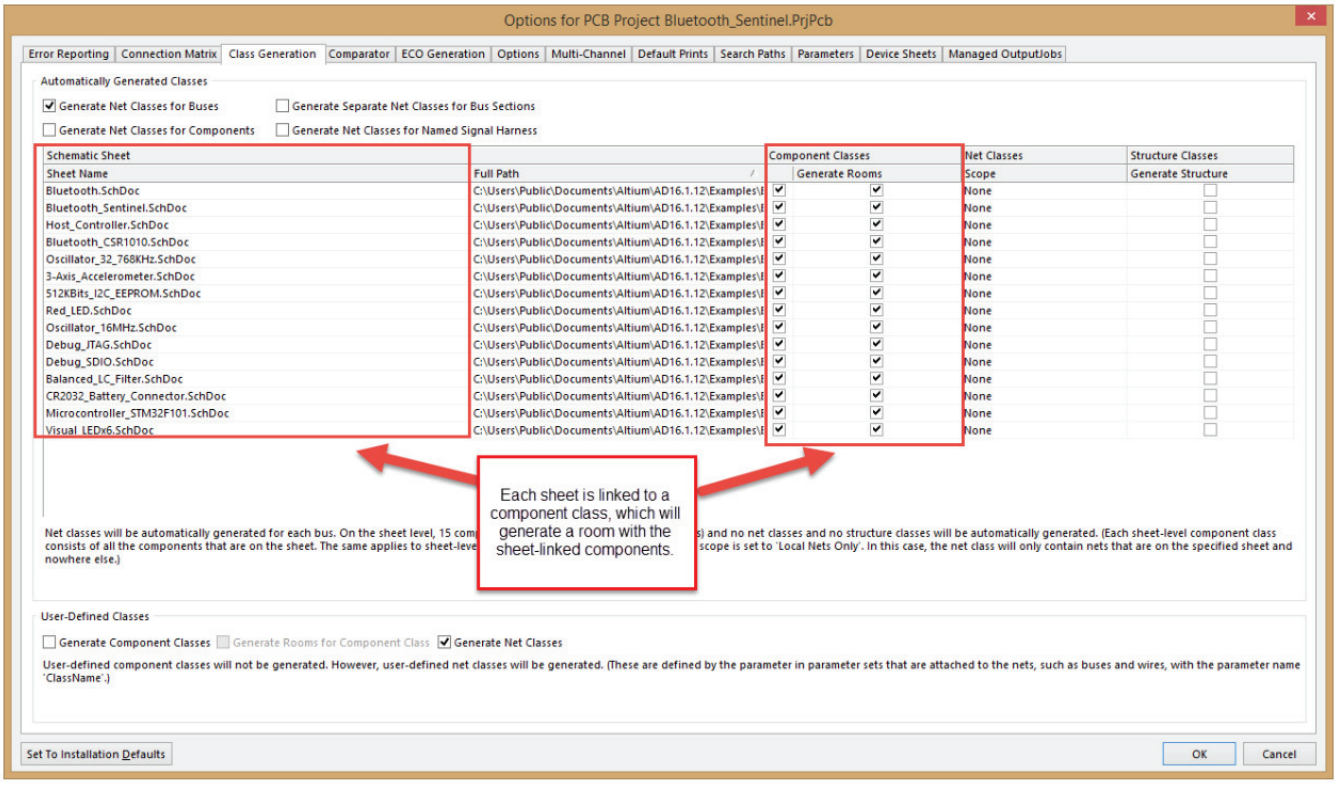


Figure 1: Generating a room with the sheet-linked components.

GROUP COMPONENTS INTO ROOMS FOR MORE EFFICIENT LAYOUT

The neat thing about rooms is the room definition's preference to allow room and component locking. With components locked within a room, shown in Figure 2, you can relocate the room and bring all the assigned components along with it in one mouse-drag action, and then keep the room stationary using room locking. This removes a user's hassle of manually moving each individual object, or a selection of a group of objects. Of course, the added capability to unlock the components to individually relocate each one is an option too, making the tool less restricted in case modifications to one object may be required.

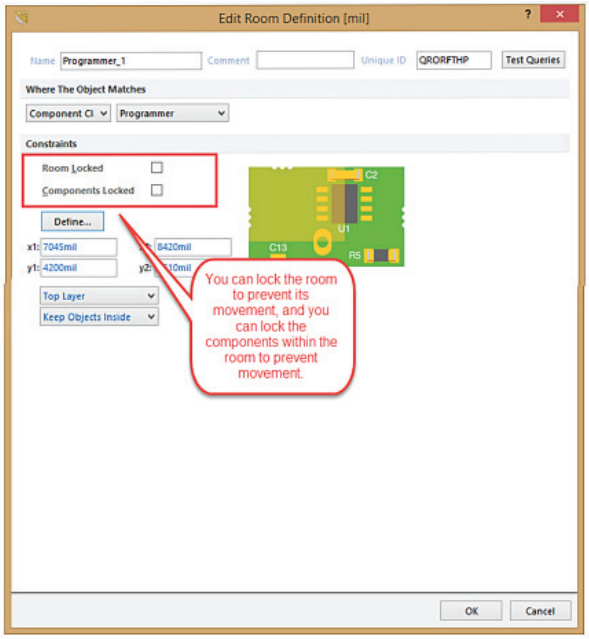


Figure 2: Rooms can be locked to prevent moving.

HIDING NETS

User-defined nets are assigned to specific objects within the PCB layout, defining the connections to be made. For example, BGAs have multiple vias and pads encompassing various nets waiting to be connected to other objects within the layout. When the BGA is left unrouted, a rat's nest of connections will appear, which causes a visual confusion on component spacing and disrupts component placements. This can be easily resolved by hiding them, which can be performed for specific nets, components, or both.

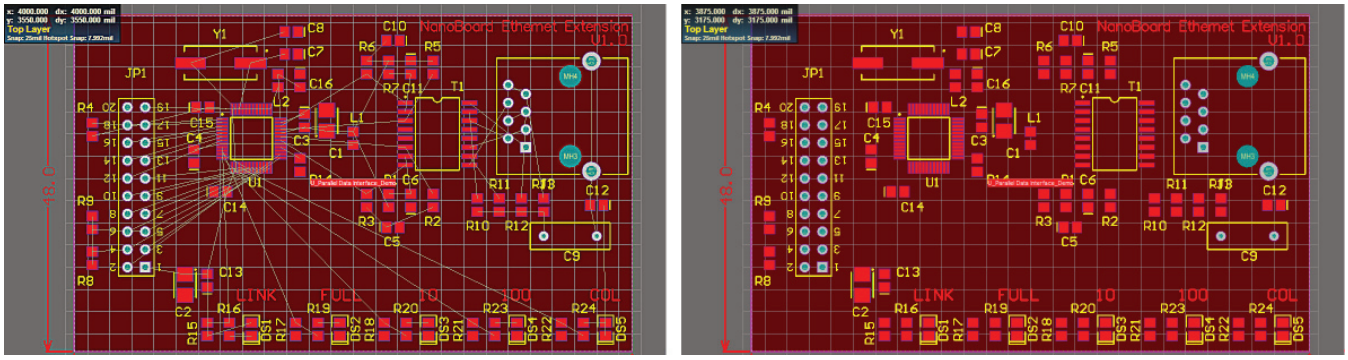


Figure 3: Hiding the unrouted package allows you to see what you are routing without the rat's nest of nets.

CONCLUSION

In summary, the utilization of rooms and appearances within a board layout can simplify your layout of components. If the tool didn't contain such features, you would spend a large amount of time laying out a board, and possibly result in not meet the deadlines. Who would have thought that colors and grouping can make a large difference to a component layout?

REFERENCES:

http://techdocs.altium.com/display/ADRR/PCB_Obj-Connection((Connection))_AD

http://techdocs.altium.com/display/ADRR/PCB_Dlg-ConfinementRule_Frame((Room+Definition))_AD