Construction of electronic systems

Exercise 5: USB DAQ project
PCB shape

Some hints

Using snapping / reference points

In order to align the PCB with the virtual enclosure, add some *reference points* in the 3D view and then use them in the 2D view for alignment.

A word of caution here: these snapping points made in the 3D view won't work in the 2D view if you have hidden the layers containing the 3D bodies where the snapping points were made! An Altium "feature" that can cause you severe headaches if you do not know about it.

Also, ***make sure that the reference snap points are actually visible*** – and not hidden! Go to "View configuration" (press "L") and scroll down to "System Colors". There check the visibility settings for the snap points (see the figure below).
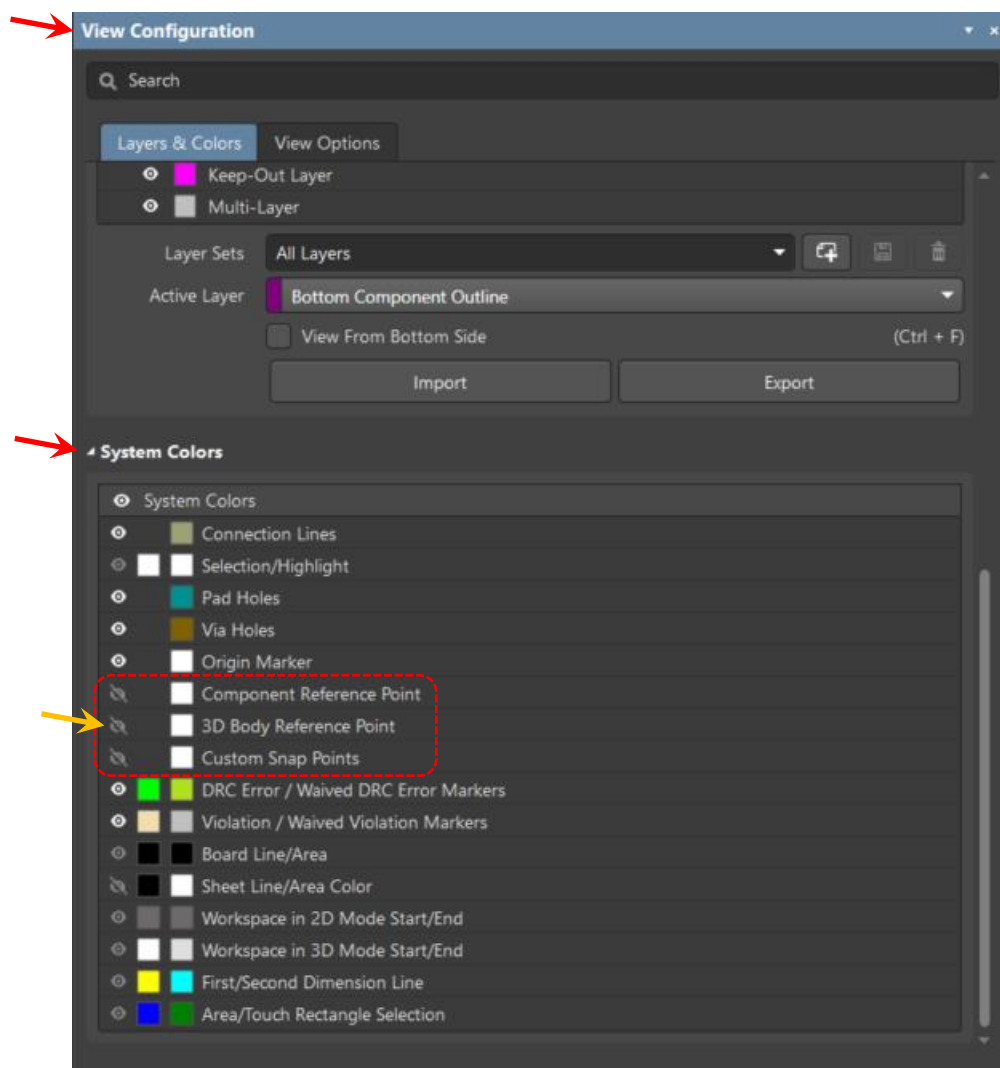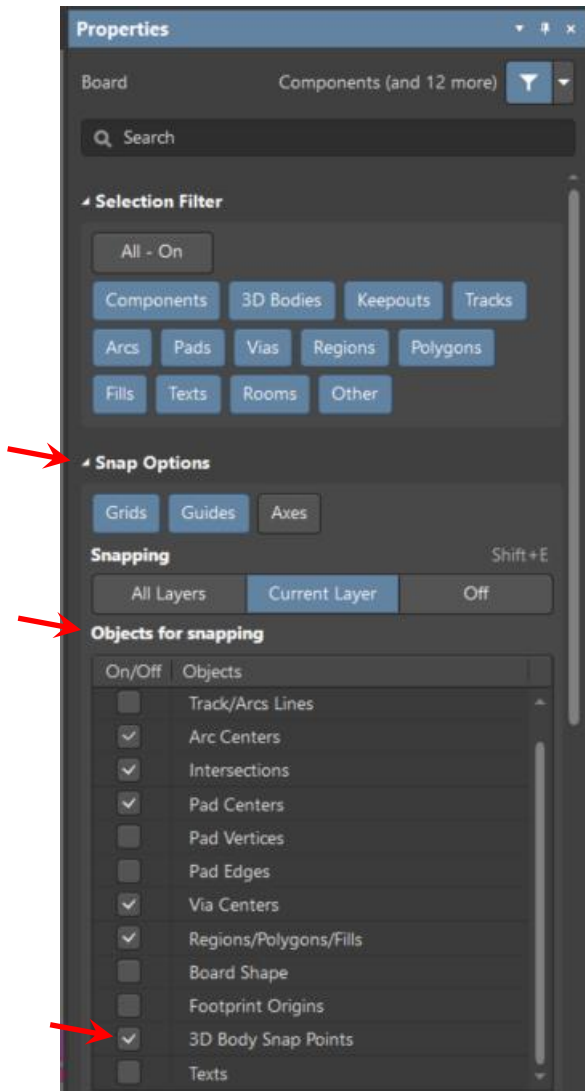

Figure 1 - if snap points are hidden, click on the "eye icon" on the left side to turn on the visibilty (yellow arrow)

Choose which snapping points are active

A reminder: in the "general properties" of the PCB design you can determine which snapping points are active. Make sure that you have the "3D Body Snap Points" selected (see below).

## How to align the PCB with the standoffs?

To make sure that the PCB really sits on the prepared standoffs, go to 3D view and check the

"Tools → 3D Body Placement → Align Face With Board"

command in the menu. The shorter way is to press the following keys: **TBF**.

With this tool you can specify onto which surface the PCB must be placed and the Altium sets the appropriate height of the 3D body containing that surface.

Note: in some cases the TBF function flips the bottom enclosure upside down. To prevent this, make sure that the "Board Side" property of the enclosure 3D model is set to "Bottom".
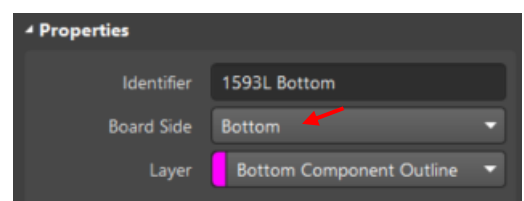
Making the mounting holes non-plated

It is a good practice to make holes where PCB is mounted to the enclosure with screws non-plated (i.e. not metalized). Why is that so? Because if the mounting hole is plated, then the diameter of the hole is reduced slightly. And that can cause an annoying situation when you want to assemble your product and you realize that the mounting screws won't fit in the mounting holes.

And besides that, the threads of the screws can damage the plating, causing it to chip away, which is also something that you would usually try to avoid.
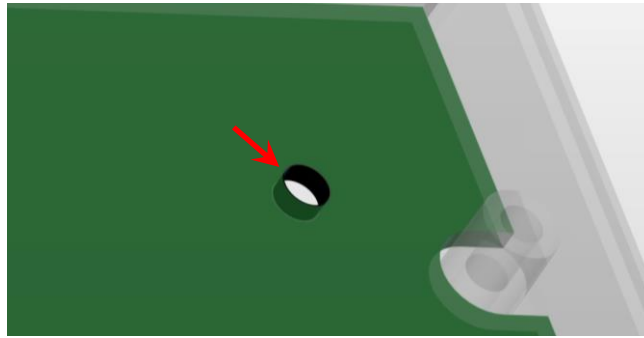

Figure 2 - make sure that mounting holes are non-plated

And where in Altium Designer you specify the type of plating for a hole? Check the properties of the hole and search the section that defines the hole geometry (see below).
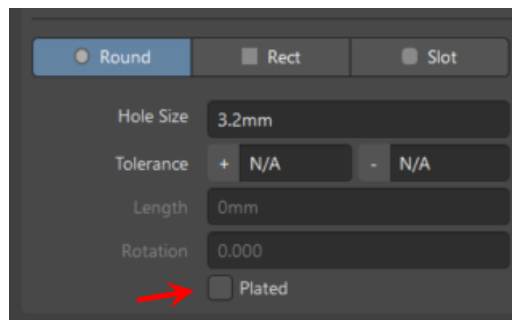

Figure 3 - the plating option for the hole

Hiding the enclosure 3D models in 2D view

Once you manage to assemble the 3D model of the enclosure, it is useful if you can hide the 3D models of the enclosure *while you are working in the 2D view*. Otherwise your view is burdened with the large hatched rectangles which "remind you" that you are working with 3D models in this PCB design. And these large hatched rectangles are really annoying when you start designing the PCB which lies inside the enclosure – that is, inside the large hatched rectangle. See the situation below.
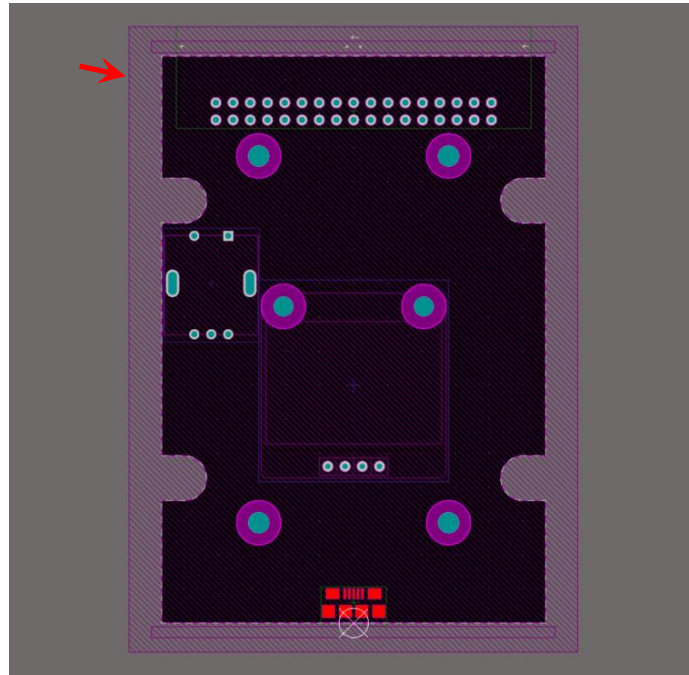

Figure 4 – in 2D view, a large hatched rectangle that represents the bottom part of the enclosure can really be annoying when you start working on your PCB design. It would make sense to hide it somehow.

It would definitely make sense if we could somehow hide this 3D model while in 2D view. How can we do that?

Well, there are several options, but here is a simple one:

1. place the 3D models of the enclosure on a "Bottom Component Outline" layer (mechanical layer 14). This makes sense, since we are mainly dealing with the bottom parts of the enclosure.

2. And now hide the "Bottom Component Outline" layer! The large hatched rectangle will obviously disappear. And since the "Bottom Component Outline" layer is not something we really need during the PCB design, we are OK with hiding it from view.

Following the idea described above, you get the following more pleasant situation below.
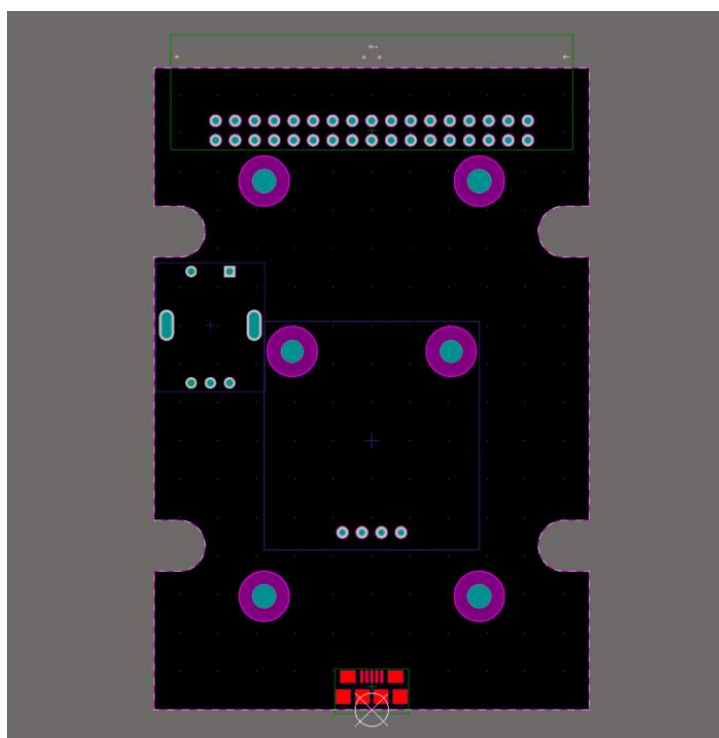

Figure 5 – once the 3D model of the enclosure is hidden, you can continue with the PCB design more conveniently