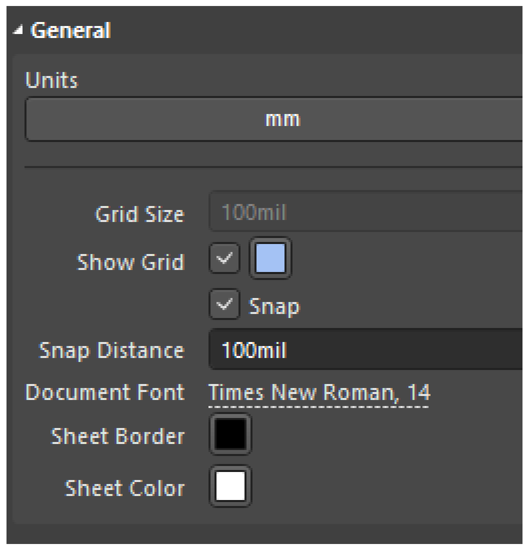
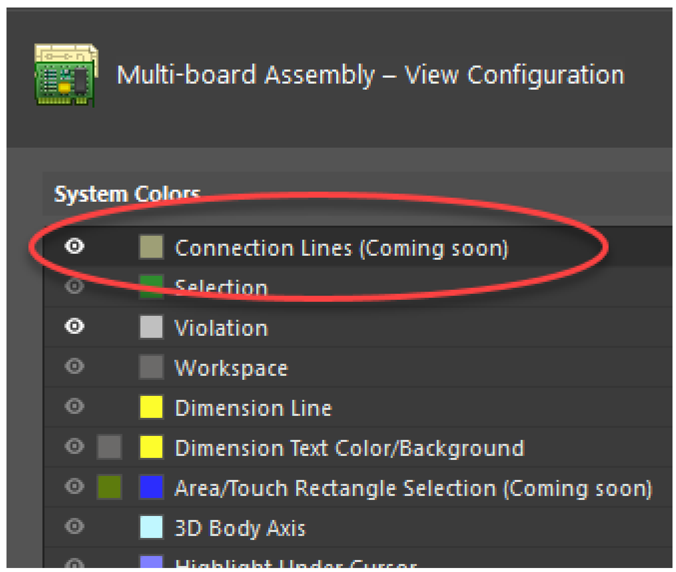
Altium Designer Multi-board Webinar Q & A

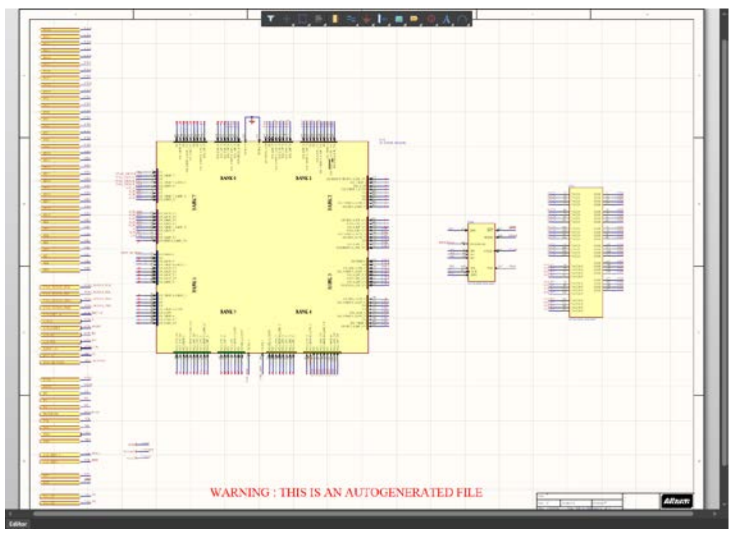
<table>
<tr><th>Question / Comment</th><th>Answer / Response</th></tr>
<tr><td>Is there any preparation work that must be done in the PCB project prior to using the project in a multi-board schematic?</td><td>Any connector in a PCB schematic that needs to be shown in the multi-board schematic needs to have the following parameter:

Name: System
Value: Component</td></tr>
<tr><td>Can the multi-schematic grid be changed to dots vs. lines?</td><td>The color can be changed, but it does not give the option for the dots.

</td></tr>
<tr><td>What precautions do you need to make in your Connectors library on pin number to make it work? If Pin 1 is not oriented the same is there an easy way to reverse it?

Can the interconnection between PCBs take account of "mirrored" connector pinning?</td><td>At this point in time (AD18.0.12), the direct connect and the wire connectivity do not have the ability to swap pins. Bug Crunch #9267 has been submitted to address this issue. It is noted that the cable and harness primitives do allow for a swapping. This switching is done in the properties of the harness and cable, not in the connection manager.</td></tr>
<tr><td>Can you adjust height in stack using real numbers like the 3D window in PCB layout? i.e. standoff, total height.</td><td>If we are interpreting the question correctly, this cannot be accomplished since there are no properties associated to any of the objects that are visually represented in the assembly.</td></tr>
</table>

Can a multi-board project contain a multi-board project? Can I make a multi-board project out of existing designs that interconnect at a higher-level assembly?	In the assembly editor, one can bring in another multi-board representation. However, this does seem to be the case in the schematic editor of the multi-board project. In an experiment using the RangeFinder multi-board schematic from the webinar, there was an attempt to bring the RangeFinder multi-board into another multi-board schematic sheet which was called 'MDB in MDB.' The module primitive allowed for it; however, the entry primitive was an issue. There was an attempt to add the 'system = component' to one of the existing connectors in the RangFinder MDB project. However, when using the command to import the entry information via ECO, Altium claimed that no changes were detected. In short, this could be a viable feature in the future with some additional work.
Is there a way to set the offsets between planes when aligning?	There is no text entry for a number or primitive to establish the offset. All of this is manipulated through the mouse actions or by using the alignment features in the tool menu.
Can you use a 3D Mouse to do this maneuvering? (Connexion, etc.)	Not as of the 18.0.12 revision. However, it is being high sought after. Please refer to Feature Request #9151 (which has been combined with #8789)
Can you use both axis-to-axis and plane-to-plane to make sure two objects are perfectly aligned?	This can be done, especially if the alignment has circular points for the axis-to-axis alignment feature. For example, one can first use the axis-to-axis to center the alignment followed by using the plane-to-plane for the plane of the two objects.
Can you export a BOM with all the added mechanical parts / standoffs, etc.? Does the MB design then compile an overall BOM?	Yes, however, note that the BOM capabilities are a bit limited and Altium will need to add some additional capabilities to make this a more viable editor.
Can you automatically check for collisions?	There is no online-DRC for collision checking. It must be invoked using Tools » Check Collisions. To clear the highlighted findings, Tools » Clear Violations
When saving/exporting PCB models in AD18, do the designators/text show up? Does multi-board PCB have option to export STEP file for mechanical engineers to look at?	There is no export feature! This was a glaring oversight by Altium in version 18.0.12. Please vote for the capability to be added in: #8798.

I have a design with two schematic sheets. I have a connector configured as multi-part - with pins for P1 on both pages. when updating an entity, it creates TWO entries, one for each schematic sheet	This is bug. Though it is represented as a multi-part component on a PCB schematic, the multi-board schematic is supposed to represent this connector as a single connector.
Assume that a given assembly can be used in multiple MBDs?	Yes, in the assembly editor, one can import another multi-board.
Is there any intelligence in the 3D assembly in terms of connections (e.g. does pin 1 on pcb1 match with pin 1 on pcb2)? Does multi-board do electrical rules check if the pins are described correctly? In multi-board assembly, does Altium give errors if a pin from one board is not matched correctly to a socket on another board as defined in the multi-board schematic?	Not in version 18.0.12. However, in looking at the system preference for the assembly editor, there's a hint that this may be forthcoming. 
It would have been nice if Altium MB would have included features which allow for the capture of cable components, such as wire, plug housings and contacts.	This is forthcoming; however, one has to keep in mind that this is not a trivial undertaking. There will need to be primitives for the assembly schematic for this to be handled. More importantly, the users will now have to create a library for these components. Would this be a schematic library or a component library? There are many concepts to consider.

Say I make Aduino based products, would you set up the multi-board with a blank aux board and then design the aux board? Would the aux board creation appear in real time to the multi-board assembly? In the multi-board schematic, you demonstrated a bottom up approach (boards first then system) have any thoughts on top down in designing the system?	The concept using the Aduino board could be done with the caveat that the PCBs drive the multi-board design. As demonstrated in the webinar, one can move footprints in either the PCB editor or the assembly editor. As for the top-down concept, multi-board as of 18.0.12 has only a bottom-up design capability. Altium would need to have a mechanism that pushes not only the net information to the PCB schematic, it would also need a mechanism that would allow the multi-board to declare a connector for the PCB schematics for which these nets would be attached. In fact, Altium had an FPGA to schematic wizard that would take the FPGA pin information and the desired FPGA symbol and place it in the schematic. This top down for multi-board concept would be comparable. If you look at the old spirit level board (SpiritLevel-SL1) that Altium provides in the examples directory, the SL_FPGA_Auto_2E.SchDoc was completely generated using this FPGA-to-schematic wizard. 
It would have been nice if Altium MB would have included features which allow for the capture of cable components, such as wire, plug housings and contacts.	This is forthcoming; however, one has to keep in mind that this is not a trivial undertaking. There will need to be primitives for the assembly schematic for this to be handled. More importantly, the users will now have to create a library for these components. Would this be a schematic library or a component library? There are many concepts to consider.

Can you import cable models into the assembly? What about capability for doing cable assembly drawings from the multi-board schematic? Can cable models be generated from harness definitions? Are the wires and cables represented in the 3D view? If I make a cable, do I have to make a module? And if so, then do I need to make a separate STEP file for it?	As of 18.0.12, the cabling capability is limited to what was shown in the webinar. Graphical representations of cabling are very limited. For example, during a webinar, a harness was quickly pulled together with real components in the multi-board schematic editor. However, during the ECO process to the assembly editor, we did not see a graphical harness representation added to the assembly editor. One could create the cable representation in a PCB schematic and then port it over to the multi-board schematic; Unless the physical cable was somehow drawn in the PCB editor, there is nothing to import into the assembly editor. One could import a STEP file into the assembly editor that represents the cable, but there will not be any relation to the multi-board schematic module. More so, there is no way to bend or manipulate the cable.
Your boards were unrouted. This tool is used to place parts before routing a multi-board design correct?	Yes, these boards were unrouted for the purposes of showing the ability to move components in the assembly editor. More so, the updates between the assembly editor and the PCB editor (and vice versa) were near real-time updates.
Can you import the relative locations of the boards from a program like SOLIDWORKS CAD rather than manipulating the locations in Altium?	This simply cannot be done because there is no universal definition of origin between software tools. In fact, this is the very reason components need to be rotated and flipped when they are imported.
It was noted that mating connectors should have the same names. But plugs fit into jacks and we designate them as J1 and P1. Will I need to change these to match one another?	You do not need to have the same connector names. However, if you wanted to have the same names for your connectors, you cannot make this modification in the multi-board schematic for the direct connect or wiring connectivity type in version 18.0.12. Rather, you will have to make the change in the PCB project and then push it to the multi-board via ECO. Cables and harnesses are a bit different since the connectors are declared in the multi-board.
What's the main benefits of using this multi-board feature [in the assembly editor]? Is it to do a clearance check?	The main benefit is to check placement and move components for boards that will be in tight quarters. Keep in mind that a mechanical chassis could also be brought in for fit check.

Combining BOMs, with multiple C1 capacitors would be only marginally useful.	Check out the following comment by Nine Dot Connects on the Altium forum regarding this issue: https://forum.live.altium.com/#posts/227693/675614 Since we are on the topic of the active BOM for the multi-board: 1. It is great that one can see all of the components for the entire project; however, it would also be useful if one could see just a BOM based off of the modules, connectors and STEPs that were brought into the design. In many companies, they would not order a kit for a design in which they would being buying everything down to the last resistor. A kit at the multi-board level would contain a line item for each PCB board, (though the option for showing the individual components as an indented heading would be useful.) For example, we have a project: • MEGA Arduino board • Custom PCB • 2 Stand Offs • 2 Screws for the stand offs • 2 washers for the stand offs. • LED screen • A temperature sensor • A distance sensor • A connector between the LED screen and the connector on the board It would be okay for me to show all of the components of our custom board; however, I would treat the board as a separate assembly with its own product number. 2. Can you please add a column that shows which project the component is coming from (if you are going to display all of the components)? There is a column that can be enabled, but it shows the entire path of the project. 3. The STEP files brought into the assembly editor are added to the BOM. Can they be shown as individual line items? More so, is there a way to provide some details to them? I cannot click on them in the assembly to get to their properties.

The "helpful" labels on the cables, wires, etc. just get in the way and make the schematic messy. Can they be eliminated?	The label cannot be eliminated; however, they can be made invisible by turning them off in the properties panel.