



ALTium **365**

Altium Designer

Essentials Course - Altium 365

Module 29: Design Review and
Commenting

Software, documentation and related materials:

Copyright © 2024 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.

ACTIVEBOM®, ActiveRoute®, A365™, Altium 365®, Altium Concord™, Altium Concord Pro™, Altium Designer®, AD™, Altium NEXUS®, Altium OnTrack™, Altium Vault®, Autotrax®, Camtastic®, Ciiva™, CIIVA SMARTPARTS®, CircuitMaker®, CircuitStudio®, Common Parts Library™, Concord™, Concord Pro®, Draftsman®, Dream, Design, Deliver®, DXP™, Easytrax®, EE Concierge®, Fearless HDI™, Geppetto®, Gumstix®, Learn, Connect, Get Inspired™, NanoBoard®, NATIVE 3D™, OCTOMYZE®, Octopart®, OnTrack™, Overo®, P-CAD®, PCBWORKS®, PDN Analyzer™, Protel®, Situs®, SmartParts™, Upverter®, X2®, XSignals® and their respective logos are trademarks or registered trademarks of Altium LLC or its affiliated companies. 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

Module 29: Design Review and Commenting	3
1.1 Purpose	3
1.2 Shortcuts	3
1.3 Preparation – Part A	4
1.3.1 Review inside Altium Designer	4
1.3.1.1 Comments and Task panel	4
1.3.1.2 See Comments and Tasks from the Trainer	5
1.3.1.3 Place Tasks and Comments	7
1.3.1.4 Placed Tasks and Comments	8
1.3.2 Review with Web Interface.....	10
1.3.2.1 Open Web Browser	10
1.4 Preparation - Part B	11
1.5 Redesign After Receiving Comments	12
1.5.1 Replace Via with Component Mounting Hole.....	12
1.5.2 Additional Comments added to your design	12

Module 29: Design Review and Commenting

1.1 Purpose



In this module, you'll learn how to open a Project for a review, using the Comments and Task panel, adding comments and collaborate with other team members.

Altium Designer projects, when integrated with a connected Workspace, enable a dispersed design team to collaboratively work on projects that are centrally stored and managed through that Workspace. To enhance the design collaboration process, the design interface supports commenting on design documents. Comments, i.e., user-added notes, can be applied to a point, object, or area on a schematic, 2D PCB, Draftsman, or ActiveBOM document and may be replied to by other users.

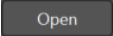
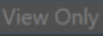
1.2 Shortcuts



Shortcuts when working with Module 29: Design Review and Commenting

Ctrl+ALT+C:	Place a Comment
Ctrl+S:	Save Document

1.3 Preparation – Part A

1. **Close all existing projects and documents.**
2. Open the Project Module 29 Design Review and Commenting – Part A
 - a) Select **File » Open Project** to open the *Open Project* dialog.
 - b) Navigate to the predefined Training Project Module 29 Design Review and Commenting – Part A
(Top\Projects\Altium Designer Essentials Training Course\...)
 - c) Select **Open** 
 - d) The Project will be available at your Project Panel with **View Only** option .

1.3.1 Review inside Altium Designer

1.3.1.1 Comments and Task panel

3. Open all schematic documents and the PCB with the command **Open All**.
4. Open the Panel *Comments and Tasks*, Figure 1, no Comments and Tasks are added so far.

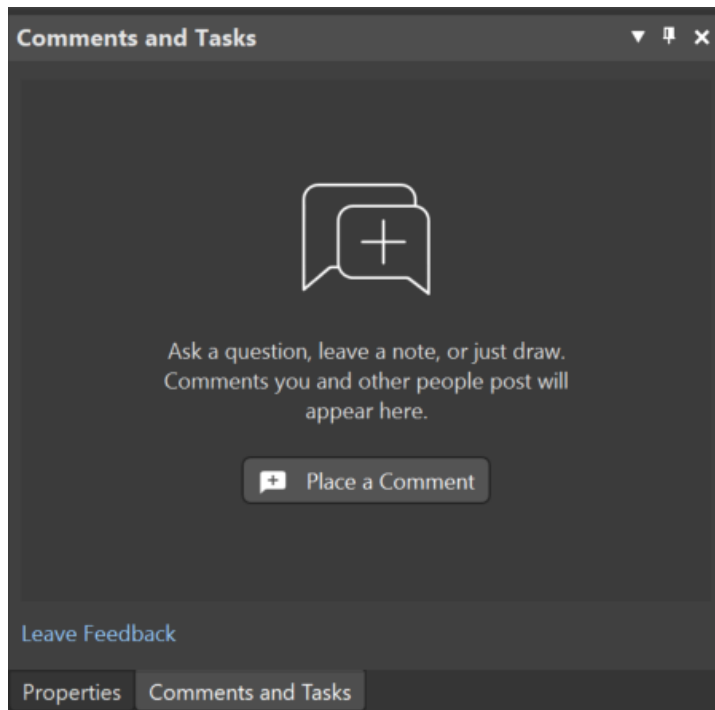


Figure 1. Comments and Tasks

1.3.1.2 See Comments and Tasks from the Trainer



For ILT Training: Wait until the trainer added some comments / changes to the project and let you know that he is ready before you continue with the next steps.

5. If you check the upper left corner of the workspace you can see icons representing other persons that opened the project / document, Figure 2.

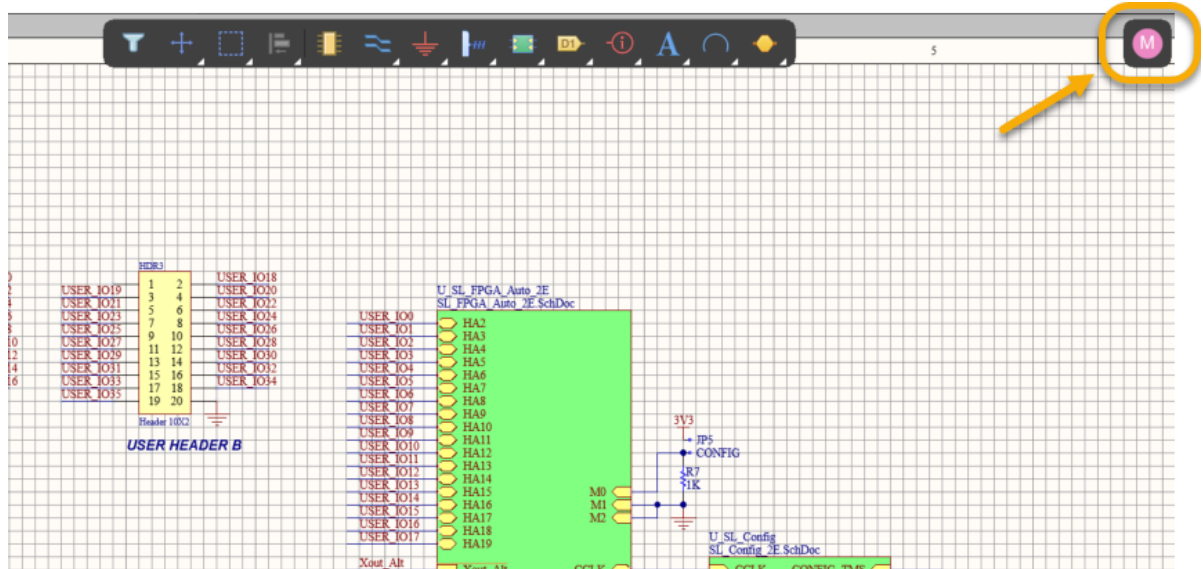



Figure 2. Other Users that opened the Document / Project

6. If other users modify the document the icon change and you see a pen, , Figure 3

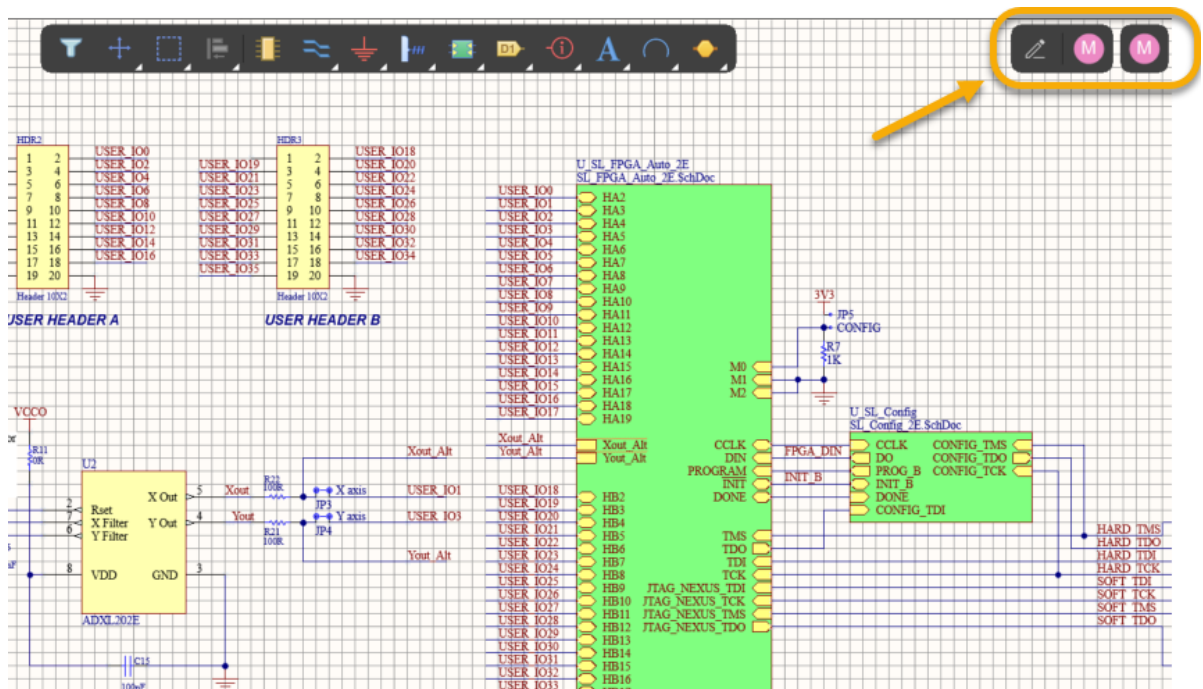


Figure 3. Document with Modification

7. Comments and Tasks that were added to the project are listed at the *Comments and Tasks* panel. Each comment and task have his own information area. This can be selected to jump to the Object in the document and to see details for the information.

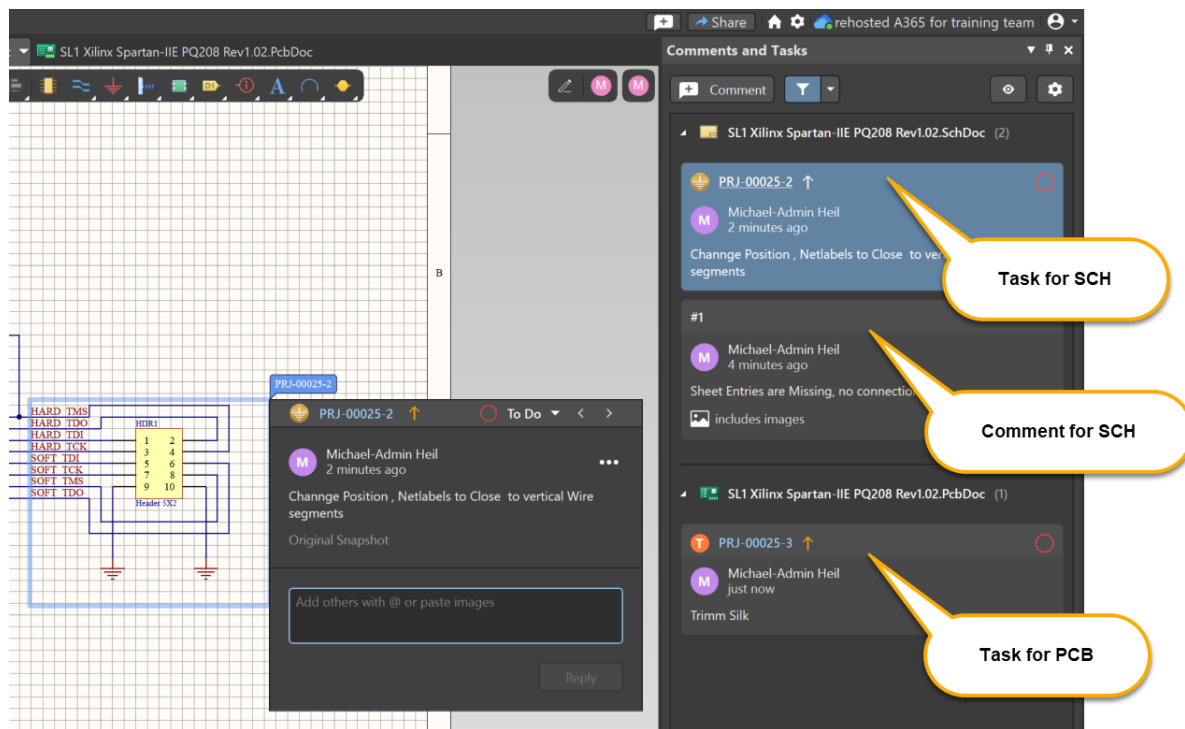



Figure 4. Notes and Comments

1.3.1.3 Place Tasks and Comments

Next you can place **ONE** comment. To place a Comment, click the **Comment** command and select an area or an object at the SCH (e.g. Component / Pin / Net), or you select at the PCB a PCB area or object and create a comment.

8. At the *Comments and Tasks* panel select the command to place a Comment .
9. Place **ONE** comment at a SCHDOC or the PCB. Feel free to assign a task to a Team Member /College or the Trainer.

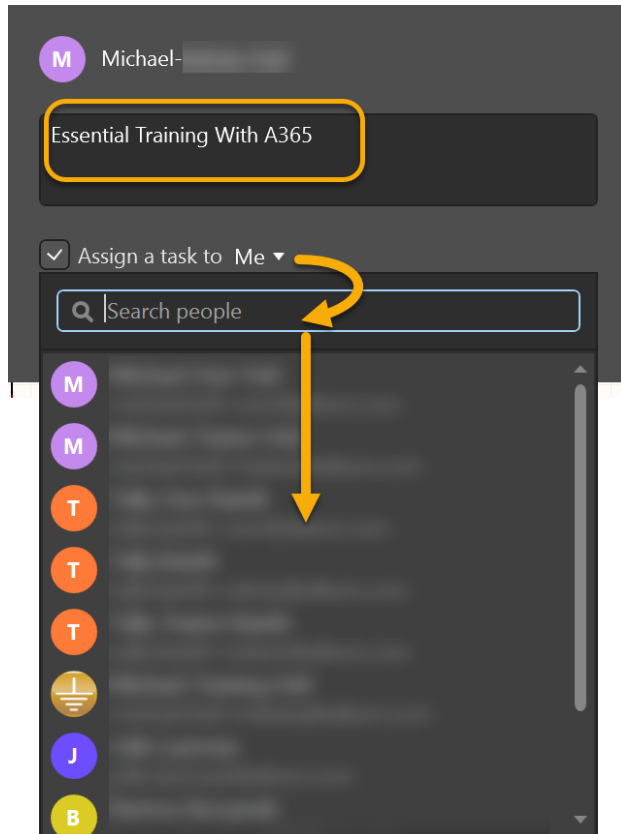


Figure 5. Adding a comment, with Task

10. If the panel was empty, after you placed your first comment you can see additional commands at the panel, Figure 6.

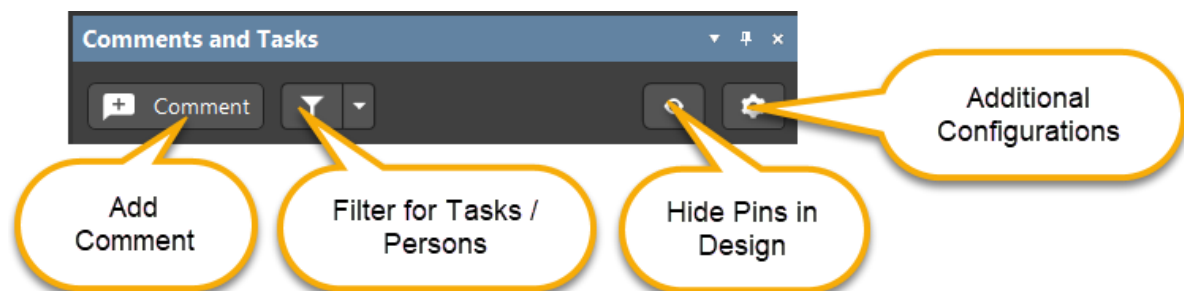




Figure 6. Commands from Comments and Tasks panel

11. You will see how the comments, task list grows, as other Training attendees place additional comments, tasks.
12. You can check if you have a task if you filter the task list.

1.3.1.4 Placed Tasks and Comments



For ILT Training: Wait until some of the attendees created some comments / tasks at the project before you continue with the next steps.

13. At the *Comments and Tasks* Panel select one of the Tasks or Comments that were added to the Project.
14. A task offers the option to change the priority and to change the status of the task. It is also possible to add additional comments to an existing task / comment, Figure 7.
15. Feel free to open the configuration  and the filter  for the *Comments and Tasks* panel to configure the list of comments / tasks you see at the panel, Figure 8

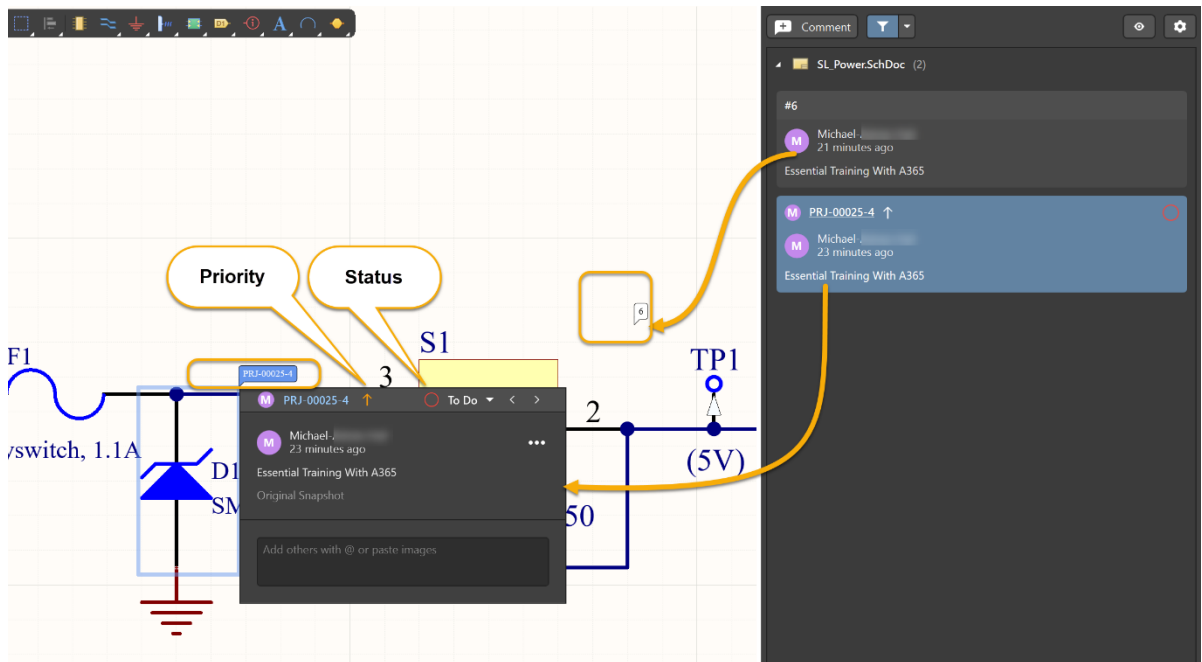


Figure 7. Open placed Tasks and Comments

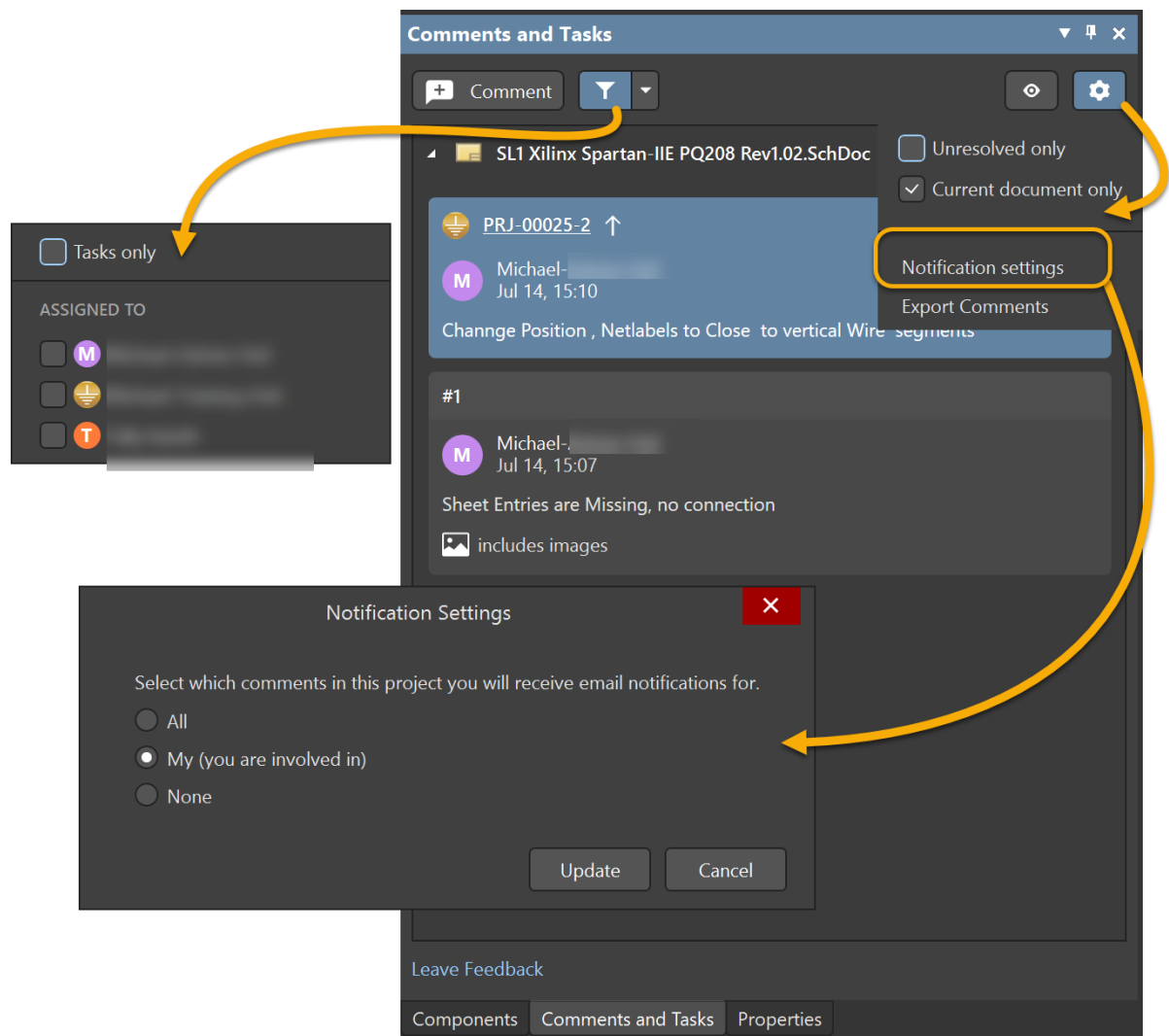


Figure 8. Configuration for the Comments and Tasks panel

1.3.2 Review with Web Interface

As alternative to a review in Altium Designer it is possible to open and review a Project from a Web Browser.

1.3.2.1 Open Web Browser

To make it simple we now open the Web Browser from within Altium Designer, but feel free to use the other methods you learned at the beginning of this Course.

16. At the *Project* panel, select the Project file, right click, and execute **Show in Web Browser**, Figure 9.

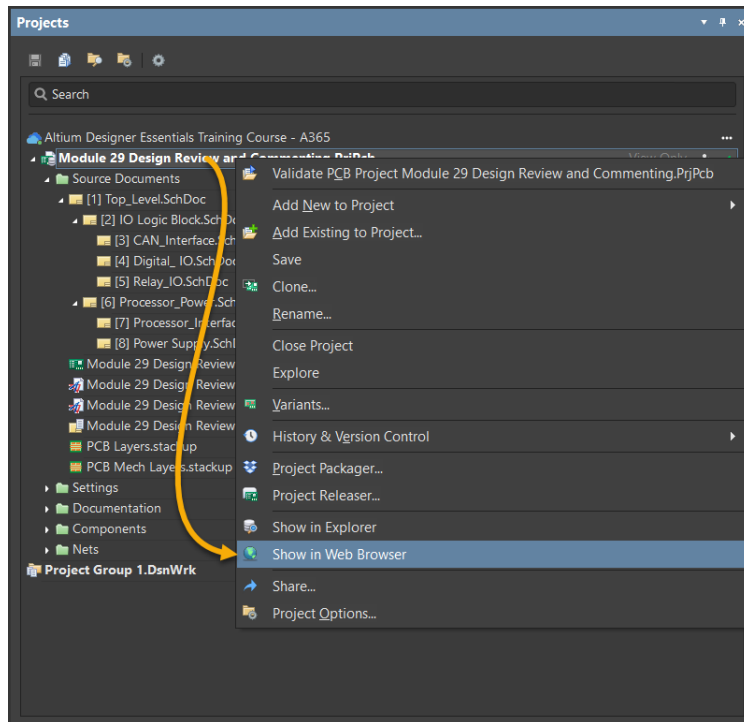


Figure 9. Open Web Browser

17. With the Web Browser open and the Project loaded:

- Open the *Comments and Task* list to see the existing Comments and Tasks
- Place **ONE** Comment at the Web View (SCH or PCB).

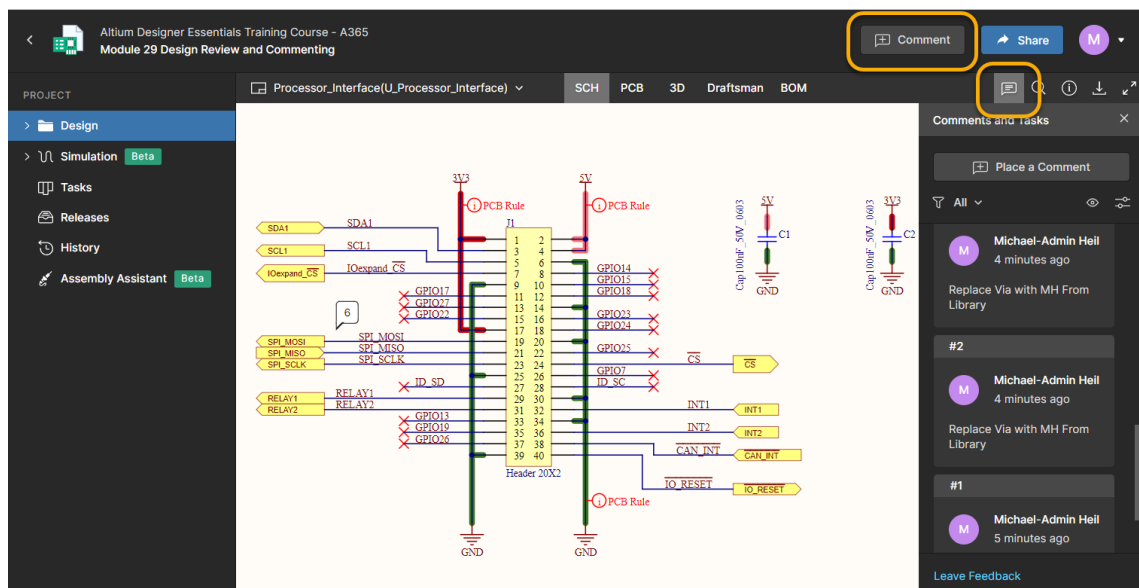
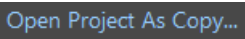



Figure 10. Web View for adding Comments

1.4 Preparation - Part B

18. **Close all existing projects and documents.**
19. Next, create a Copy / Clone of the Training Project Module 29 Design Review and Commenting - Part B
20. Select **File » Open Project...** to open the *Open Project* dialog.
21. Navigate to the predefined Training Project Module 29 Design Review and Commenting - Part B
(Top\Projects\Altium Designer Essentials Training Course\...)
22. Select **Open Project as Copy...** 
23. At the new dialog *Create Project Copy*
24. Add your name to the project: to Module 29 Design Review and Commenting - Part B - [Your Name]
 - a) Open the *Advanced* section.
 - b) 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**
 - c) Change the Local Storage path if needed.
 - d) Select **OK** to create the copy.
25. Wait until Altium Designer creates the copy of the project and opens the project in the *Projects* panel; this can 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.

1.5 Redesign After Receiving Comments



For ILT Training: Wait until the Trainer added some comments / tasks at the project before you continue with the next steps.

Now your Trainer will add comments to your design 29 Design Review and Commenting – [Your Name] . Read the Comments and Update your Design based on the Comments you see.

One Comment / Task is identical for all attendees - Replace the Mounting Hole Via with the Component Mounting Hole, the other Comment / task will be random but will cover a Topic that you have seen during the training.

1.5.1 Replace Via with Component Mounting Hole

For the training project the mounting holes are created from Vias, a better approach is to use Pads, or mounting mole components.

Next you will find a general workflow to replace the vias with the mounting hole component.

26. Place the component Mounting Hole 620-275 Plated four times in the Top Sheet.

Feel free to add a note with additional Information.

27. Annotate the design so that the mounting holes have unique designators.

28. Transfer the netlist to the PCB, update PCB document

29. Place the footprint for the mounting holes, remove the vias that were used as mounting holes.

30. Check the PCB by running the DRC

31. If the DRC detects errors, resolve the errors before moving to the next step.

1.5.2 Additional Comments added to your design

32. Check your design for additional comments that were added to your design.

33. Update your design based on the comments you see from the Trainer from your design

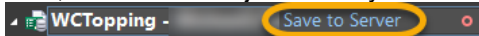
34. Update the comments / tasks for your design.

When ready, save all modifications and save the modifications to the server.

35. Save all documents using **File » Save All**.

36. 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 29: Design Review and Commenting – [Add Your Name] – Finished*.

iii) Select **OK**.

37. When ready, close the project and any open documents, **Window » Close All**

Congratulations on completing the Module

Module 29: Design Review and Commenting

from the

Altium Designer Essential Course

Thank you for choosing Altium Designer