Totally not ripped from Hyperloop

**TortoiseSVN & SVN Version Control with Altium**

We have version control with Altium now!! This is great, because we can now collaboratively work on Altium schematics and PCBs without having to wait for others’ changes or cumbersomely uploading to google drive to share files.

1. Launch TortoiseSVN from the CAEN software tab that opens on startup. Make sure you launch it from the “Cloudpaging Player” app that opens as well.
2. Launch Altium
3. Open up the file browser and navigate to your Documents folder. Right click and select TortoiseSVN > Settings. From the dialog that opens go to the “Network” panel on the left side.
4. The bottom field “SSH Client” should be empty. Click Browse. In the dialog, click the Windows (C:) drive on the left side and then in the file browser navigate to VApps > TortoiseSVN > [arbitrary folder name] > bin > TortoisePlink.exe. Click Open.
5. **This step is very important!** The “SSH Client” field should have been populated with a file path to the TortoisePlink executable. After this file path, type a space and then the following characters:   
     
    -l uniqname -pw password  
     
   Fill in your uniqname and password in the text above. This step makes it so you don’t have to enter your password every time you send an svn request.
6. Click Apply and then OK.

**Unfortunately the six steps above must be completed every time you login to CAEN.** I wish this wasn’t the case but it’s (currently) the easiest and fastest way of doing this version control business with Altium.

Your TortoiseSVN client will now be able to svn+ssh tunnel into a repository on a CAEN server. See the following steps for connecting to a remote repository. The example we will use is the “mhype-devices” repository that contains all of MHype’s custom devices and their associated schematic symbols and footprints (there is also a mhype-connectors library). **You need to slack me (Albert Anwar) before completing the following steps, because I will need to add read+link permissions for your uniqname, or else the repository checkout will fail.**

1. Select a folder you wish to checkout the repository to. This can be any arbitrary path. The TortoiseSVN client will create a new folder containing the project for you in this location.
2. Right click in this folder and select “SVN Checkout...”
3. In the dialog that opens, enter the following URL into the “URL of repository” field:  
     
   svn+ssh://uniqname@login.itd.umich.edu/afs/umich.edu/user/a/w/awanwar/mrover/pcbs/radio  
     
   Fill in your uniqname in the uniqname field. The green highlighted section will be the same for any repository you checkout (as long as the host is always the same). The yellow section will be different depending on where the repository resides. All of the Power subteam’s PCBs will reside on my account (awanwar), so you will only have to worry about the last purple highlighted section. However, if you are on Controls the repository could be hosted on Ben or Jonah’s accounts, so make sure you know the exact URL.
4. Leave the rest of the fields in this dialog alone, and click OK to checkout the repository. TortoiseSVN will access the repository and check it out into the directory.   
     
   *If this operation fails, talk to the repository owner to see if they gave you the correct permissions.*
5. You should see a new folder with the repository’s name. In this case, “custom-parts”. If you open this up you should find three folders: branches, trunk, and tags. Trunk contains the main branch of files: the most up-to-date revision. Tags and branches will contain other files that, in the case of tags, will contain older, stable versions of the project. In the case of branches, it will contain files that have diverged from trunk, perhaps a new feature that is being worked on that has not yet been finalized.

You now have a working copy of the repository on your local computer. The following steps will outline how to work with them inside of Altium.

1. Open up Altium and find the trunk folder of the repository you just checked out. Open the “MHype Devices.LibPkg” file.
2. Altium will load this project along with the associated schematic and PCB library files. You should see green check marks next to the files in the project browser panel on the left side in Altium. This means that the files are up to date and have not been modified in your local working copy.   
     
   If you don’t see green check marks on any project, this means it is out of date. **You need to right click on the project > Version Control... > Update Whole Project...** This will update changes from the server.
3. Make some changes to any part of this project, and th]
4. en save it.
5. You should now see a red circle next to the file you modified in the project browser panel in Altium. This means that the file has been modified.
6. Right-click on the file and select Version Control > Commit...
7. This will prompt you to enter a commit message. **Always enter an informative commit message!** It helps others know what you did. Then select OK.
8. This operation should fail. This is because most of you will not have write permissions on this custom library. If it does succeed, I did something wrong. **However, this modify-commit process is common among any SVN project, so the workflow here is the same as anywhere else.**
9. If you want to see the history of the project, navigate to the menu at the top of Altium and select Project > Local History > Storage Manager... This will show all commits with messages and their authors on the project.
10. To see the status of your repository compared to the one in version control,, right click on the project and select Version Control > Refresh.
11. To get actual updates from the repository, select Version Control > Update Whole Project...

**~~Currently I haven’t figured out how to handle merges & merge conflicts!~~** ~~If these occur I will handle them on a case-by-case basis.~~ False I know how to handle this

**URLs of MRover PCBs:**

**Custom Radio:**

svn+ssh://uniqname@[login.itd.umich.edu/afs/umich.edu/user/a/w/awanwar/mrover/pcbs/radio](http://login.itd.umich.edu/afs/umich.edu/user/a/w/awanwar/mrover/pcbs/radio)

**Custom Parts Library:**

svn+ssh://uniqname@login.itd.umich.edu/afs/umich.edu/user/a/w/awanwar/mrover/pcbs/mrover-parts