

Introduction to Circuit Design and Altium

ECE295 Winter 2022

November 2021

Contents

1 Getting Started	4
1.1 Altium License through U of T	4
1.2 Downloading	4
1.3 Remote Access to Altium: Login using ECF	4
1.4 License Alternative	4
1.5 Opening Altium and Getting a License	4
1.6 Project Setup	5
1.6.1 Initializing and linking to version control	6
1.6.2 Getting started with a repository your teammate has made	9
1.6.3 Resolving git conflicts with Altium files	9
2 Basic Schematic Capture	13
2.1 Placing Parts	13
2.1.1 Adding the ECE295 Custom Libraries	14
2.1.2 Manufacturer Part Search	14
2.2 Example with all parts placed	15
2.3 Wiring, Net Labels, Ports and Power Ports	16
2.4 Annotation	17
2.5 Validation	17
2.6 Adding Libraries	18
3 Pre-PCB Layout Tasks	19
3.1 Footprint Manager	19
3.2 Footprint Creation	20
3.3 Linking Part Numbers	23
3.4 BOM Generation and Validation	24
4 Creating Your PCB	26
4.1 Add a PCB Document and Define a Board Shape	26
4.2 Synchronizing with Schematic and Importing	26
4.3 Configuring PCB Manufacturing Rules	27
4.4 Layout Guidelines	28
4.5 Design Rule Check	32

5 Preparing Files to Send For Manufacturing	34
5.1 Output Job	34
5.2 Gerber Files	34
5.3 NC Drill Files	35
6 Advanced Schematic Capture	37
6.1 Symbol Generation	37
6.2 Hierarchy	38
6.3 Net Classes	38
A Guide to selecting parts from Digi-Key	40
B Adding parts to your Library from 3rd Party Websites (SnapEDA Example)	43

1 Getting Started

Let's dive right in!

1.1 Altium License through U of T

This year we now have access to an Altium Live account through U of T that will be set up for you and linked to your xxxxxxxxxxxx@mail.utoronto.ca email. You should receive an email with the subject *Your AltiumLive Account is Ready* that will provide a username and password. Log in here and navigate to **My Profile** under the user icon at the top right to change the password.

1.2 Downloading

With an Altium subscription it is straightforward to download any version of Altium from their website, here. Any version should be fine to download, but to match the version installed on the U of T lab computers, select **21.8.1** then press **Download**. Note: Altium can only be run on windows computers or virtual machines.

The screenshot shows the Altium Designer download page. On the left, there is a logo and the text "ALTIUM DESIGNER". Below it is a brief description: "Leverage the most powerful, modern and easy-to-use PCB design tool on the market. Altium Designer brings together unified design and Native 3D™ PCB capabilities to help you create next-generation electronics." A "Give Altium Designer a test run, request the free trial" button is visible. On the right, a dropdown menu shows "21.8.1" which is highlighted with a red box. Below the dropdown are "Release Date: 2021-10-21", "Installation and management", and a "Download (exe, 26.73 MB)" button, which is also highlighted with a red box.

1.3 Remote Access to Altium: Login using ECF

Altium is also available on the ECF computers along with the **BA3128** lab. You can remotely login to these computers through the ECF website here.

Use your ECF credentials to login (not your UTORID ones), and click the "Windows Remote Desktop" option at the top after logging in, from there you can choose the venue BA3128 of Communications (SF2201).

[Account Status](#) [Staples Printing](#) [Lab Status](#) [Lab Schedules](#) [Change Password](#) [Windows Remote Desktop](#) [Remote Linux](#) [Log Out](#)

Windows Remote Desktop

All ECF labs are closed due to the ongoing public health crisis.

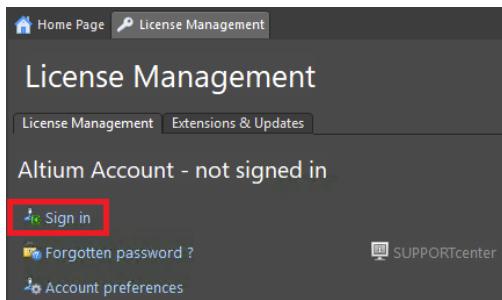
If you are logging in from off campus, you may need to use a VPN before you can connect remotely. The resources for setting up the general UofT VPN are located [here](#).

1.4 License Alternative

If there is an issue with the U of T Altium Live license, it should also be possible to get a one year student license using your UofT email. Follow this link, and fill out the form on the page. You should get access to download and install Altium within 24 hours. If you can't see the form on the linked webpage, try using a different browser.

1.5 Opening Altium and Getting a License

On startup, Altium will typically open a License page if you are not signed in from a previous session. Sign in can also be accessed from the *Home* tab that opens at startup. Select **Sign in** and enter the same email and password used at the Altium website.



After sign in, the *License Management* page will typically appear and show your allocated license with the tag **Used by me**

The U of T license is based on a number of available seats, so it is important that when you are done working you **Release** your license. To retake a license, select one that is not expired from the list and press **Use**

On Demand						
Product Name	Activation ...	Used	Assigned Seat...	Expiry	Status	Subscription St...
Altium Designer	L6WR-PAW2	-	0/50	18-Oct-2014	Expired	pired on 18-Oct-2014
Altium Designer	XBLP-R635	-	0/1	22-Nov-2016	Expired	pired on 22-Nov-2016
Altium Designer	ZHER-VV9E	-	0/1	2-Oct-2020	Expired	pired on 2-Oct-2020
Altium Designer	2MJJ-26AK	-	0/1	2-Feb-2019	Expired	pired on 2-Feb-2019
Altium Designer	DV7M-QY5N	-	0/1	21-Oct-2022	OK	Valid to 21-Oct-2022
Altium Designer	P4UG-L52E	Used by me	1/6	11-Aug-2022	OK	Valid to 11-Aug-2022
Altium Designer Custo...	TLPF-LASK	-	0/1	Imm...e	OK	pired on 30-Apr-2020
Altium Designer Custo...	74KT-BX9D	-	0/50	18-Oct-2011	Expired	pired on 18-Oct-2011
Altium Designer Custo...	9V22-DQGT	-	0/50	23-Oct-2012	Expired	pired on 23-Oct-2012

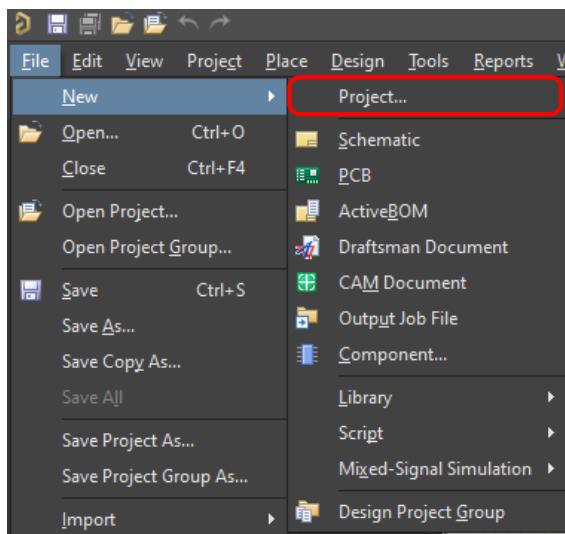
Use **Save standalone license file** **Extensions and up...**
Roam **Add standalone license file** **How to use an on...**
Release **Setup private license server** **How to use a sta...**

Note 1. When you are done working in Altium, release the license by selecting **Release** on the *License Management* tab.

1.6 Project Setup

Once you have Altium installed, it is time to start a project. Select **File → New → Project**. In Altium, a project is the name of the overarching file that manages your design. This typically includes at least a schematic and Printed Circuit Board (PCB), but it can also include other elements of the project such as mechanical drawings, and outputs that tell manufacturers how to make your PCB.

Give your project file a descriptive name and choose a location to save it locally. Once the project file is created, add a schematic to the project under either **File → New → Schematic** or by right-clicking the project file in the left-hand *Projects* panel and selecting **Add New to Project → Schematic**. It is also good to save the schematic file with a descriptive name.



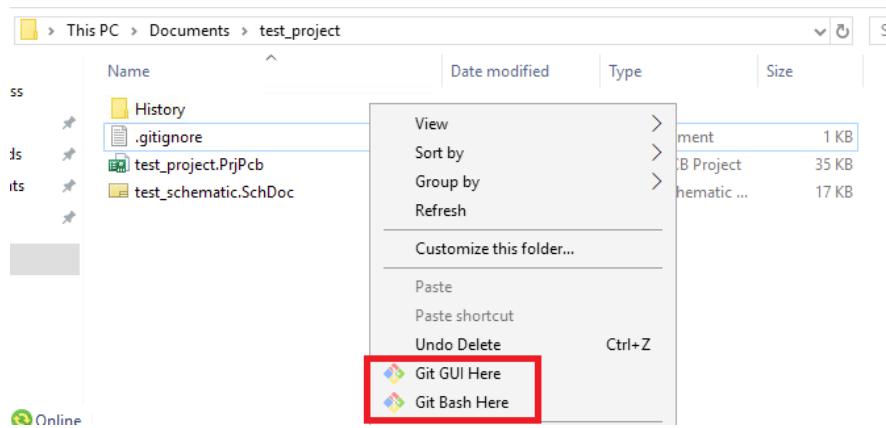
1.6.1 Initializing and linking to version control

When working on a PCB in a group it is beneficial to use version control. In ECE295, *git* is used for tracking code as well as for tracking other group files, including Altium schematics and PCBs. Altium does have tools to interface with *git*, but the initial tracking setup needs to be done through either the command line or the Git Gui tool used in ECE295.

This example shows how to link a new project (`test_project.PpjPcb`) with a schematic file (`test_schematic.SchDoc`) to the ECE295 *git* server. The project is located on the users profile of the ECF machines at This PC → Documents → `test_project`. The actual path of this on the ECF machines is `\\\VSRV1\B.Homes$\\<YOUR_UTORID>\Documents\test_project`. The project may be located anywhere, but it is important to be able to navigate to it using the command line for *git* setup.

ECE295 makes use of the Git Gui tool, which is installed by default on the ECF machines. On your own computer, if *git* is not already installed it can be found here. To add the Altium files to version control for the first time, navigate to the repository containing the files you would like to track and open a git-capable command line.

Info 1. By default, both the windows terminal and the windows powershell on the ECF machines have *git* installed. However, they do not have easy access to a text editor, which is often helpful. It is possible to run these programs from the powershell using `bash -c "nano filename.txt"` or `bash -c "vi filename.txt"`. This is still somewhat clunky, so it is also possible to use *git* to provide a bash window by right-clicking in the file manager and selecting **Git Bash Here** (see image below). Using this window the commands `nano filename.txt` and `vim filename.txt` will work. On individual windows PCs there are a variety of solutions to this. Installing *git* from the link above should provide the same right-click **Git Bash Here** functionality that is on the ECF machines, but any bash environment should resolve this. This whole thing is a non-issue on Linux and Mac, but then Altium can't run on those. Such is the life of a design engineer!



Enter the commands `git status` first to check that there is not already tracking. In general it is always good practice for your first `git` command to be `git status`.

Then enter `git init`. You should see a response indicating that the repository was initialized.

```
@BA3128WS18 MINGW64 //VSRV1/B.Homes$ /Documents/test_project
$ git status
fatal: not a git repository (or any of the parent directories): .git

@BA3128WS18 MINGW64 //VSRV1/B.Homes$ /Documents/test_project
$ git init
Initialized empty Git repository in //VSRV1/B.Homes$/Documents/test_project/.git/
```

After this, running `git status` again will show the files in the repository that are not staged for commit.

```
@BA3128WS18 MINGW64 //VSRV1/B.Homes$ /Documents/test_project (master)
$ git status
On branch master

No commits yet

Untracked files:
  (use "git add <file>..." to include in what will be committed)
    History/
    _Previews/
    test_project.PrjPcb
    test_schematic.SchDoc

nothing added to commit but untracked files present (use "git add" to track)

@BA3128WS18 MINGW64 //VSRV1/B.Homes$ /Documents/test_project (master)
$
```

At this point it is good practice to add a `.gitignore` file to ensure that only the files you really want to track are included. For an Altium project the `.gitignore` should contain the following info:

It isn't possible to make a `.gitignore` file in the file management GUI interface in windows, so instead create the file with the command `nano .gitignore`. The file needs to contain some text to be saved. To save and exit press **Ctrl+O** then **Ctrl+X**.

```
MINGW64//VSRV1/B.Homes$ nano .gitignore
History/*
__Previews/*
*.log \\
*.htm \\
*.PrjPcbStructure|
```

[Wrote 5 lines]

AG Help ^O Write Out ^W Where Is ^K Cut ^T Execute ^C Location
 ^X Exit ^R Read File ^\ Replace ^U Paste ^J Justify ^/ Go To Line

Content of .gitignore:

```
1 History/*
2 __Previews/*
3 *.log \\
4 *.htm \\
5 *.PrjPcbStructure
```

:

To include the .gitignore file in the repository to also be tracked, add it with git add .gitignore.

Commit to the newly created git repository with:

```
git commit -m "Describe the changes with a commit message here"
```

To share your work with collaborators it is important to send it to a location on the internet where they can access it, rather than just leaving it in a file on your local computer. This purpose is often fulfilled by companies like Bitbucket or GitHub, but in ECE295, git tracking takes place on your individual team repositories on U of T's ug machines.

The following steps create the remote repository on the ug machines:

```
1 # first login to the remote location and initialize a git repository with the following sequence of commands
2 > ssh <your_UTORID>@ugXXX.eecg.utoronto.ca
3 #XXX is the machine number you want to connect through, any machine from 101 to 251
4 > yes
5 > <your_ug_machine_password>
6
7 > cd /nfs/ug/groups/ECE295S/hdc-YYY #Replace YYY with your group number
8 > mkdir <name_of_git_directory>
9 > cd <name_of_git_directory>
10 > git init --bare
11 > exit #this will close your ssh connection to the ug machine
```

:

```
$ ssh [REDACTED]@ug251.eecg.utoronto.ca
[REDACTED]@ug251.eecg.utoronto.ca's password:
Linux ug251.eecg 4.19.0-18-amd64 #1 SMP Debian 4.19.208-1 (2021-09-29) x86_64

The programs included with the Debian GNU/Linux system are free software;
the exact distribution terms for each program are described in the
individual files in /usr/share/doc/*copyright.

Debian GNU/Linux comes with ABSOLUTELY NO WARRANTY, to the extent
permitted by applicable law.
Last login: Tue Jan 25 18:56:48 2022 from bras-base-toroon0259w-grc-02-184-148-156-215.dsl.bell.ca
ug251:[% cd /nfs/ug/groups/ECE295S/hdc-001
ug251:/nfs/ug/groups/ECE295S/hdc-001% mkdir altium_project
ug251:/nfs/ug/groups/ECE295S/hdc-001% cd altium_project/
ug251:/nfs/ug/groups/ECE295S/hdc-001/altium_project% git init --bare
Initialized empty Git repository in /nfs/ug/groups/ECE295S/hdc-001/altium_project/
ug251:/nfs/ug/groups/ECE295S/hdc-001/altium_project% exit
```

Back on your local machine, take the files in your repository and link them to the remote repository on the ug machines. Then set the ‘upstream’ location of the local git repository to the remote location and push your changes to the remote location.

```
1 > git remote add origin <your_username>@ugXXX.eecg.utoronto.ca:/nfs/ug/groups/ECE295S/hdc-YYY/<
   name_of_git_directory>
2 > git push --set-upstream origin master
```

Info 2. If you accidentally type the wrong thing during `git remote add origin`, you can remove it and try again with `git remote remove origin`.

At this point, the typical git work flow can be followed. For this course that should look something like this:

```
1 # sit down to start working
2 > git status #check that you don't have any uncommitted changes
3 #if there are new files that you don't want to track, add them to the .gitignore
4 > git add * #stage any changes to commit
5 > git commit -m 'detailed commit message' #commit any changes you did have
6 > git pull #get your colleagues' changes and resolve any conflicts
7 # work for a while, making changes
8 # either you finish working for the day, or finish something you've been working with for a while
9 > git status #just to check
10 > git add * #add new and modified files
11 > git commit -m 'Commit message describing the work you accomplished'
12 > git push #share work with your colleagues
```

```
ABA3128WS09 MINGW64 //VSRV1/B.Homes /Documents/test_project (master)
$ git push origin master
@ug251.eecg.utoronto.ca's password:
Enumerating objects: 5, done.
Counting objects: 100% (5/5), done.
Delta compression using up to 12 threads
Compressing objects: 100% (5/5), done.
Writing objects: 100% (5/5), 7.89 KiB | 449.00 KiB/s, done.
Total 5 (delta 0), reused 0 (delta 0), pack-reused 0
To ug251.eecg.utoronto.ca:/nfs/ug/groups/ECE295S/hdc-001/altium_test.git
 * [new branch]      master -> master
```

1.6.2 Getting started with a repository your teammate has made

If a remote repository has already been set up for something and a teammate has already committed some files there, it is straightforward to get access to those files locally and start working on them yourself.

The following commands initialize a new local git repository, link it to the already-created remote repository, and then pull the content of the remote repository to the local computer.

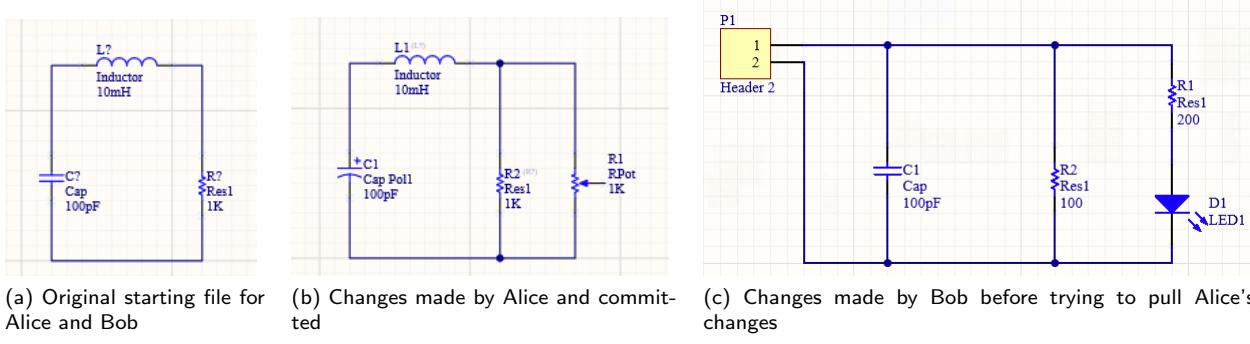
```
1 #navigate to a new folder where you want to store the local files associated with the project
2 > git init
3 > git remote add origin <your_username>@ugXXX.eecg.utoronto.ca:/nfs/ug/groups/ECE295S/hdc-YYY/<
   name_of_git_directory>
4 > git pull origin master
5 > git branch --set-upstream-to=origin/master master
```

After these steps the typical git work flow described above can be followed.

1.6.3 Resolving git conflicts with Altium files

It is very possible to come across conflicts working with Altium files and git. Say Alice commits and pushes her changes to the server. Then, Bob makes changes to the same starting file that Alice had and then tries to pull Alice’s changes. This will cause a conflict as Altium Designer files are binaries and git does not know how to cleanly merge non-text files.

Altium does have a tool to help resolve conflicts. This guide will demonstrate the work flow with a toy example.



When Bob goes to try and push his changes the push will fail. Then, when he pulls Alice's changes, a conflict will occur.

```
@BA3128WS08 MINGW64 //VSrv1/B.Homes$ cd Documents/test_project (master)
$ git push
@ug251.eecg.utoronto.ca's password:
To ug251.eecg.utoronto.ca:/nfs/ug/groups/ECE295S/hdc-001/altium_test.git
 ! [rejected]          master -> master (fetch first)
error: failed to push some refs to 'ug251.eecg.utoronto.ca:/nfs/ug/groups/ECE295S/hdc-001/altium_test.git'
hint: Updates were rejected because the remote contains work that you do
hint: not have locally. This is usually caused by another repository pushing
hint: to the same ref. You may want to first integrate the remote changes
hint: (e.g., 'git pull ...') before pushing again.
hint: See the 'Note about fast-forwards' in 'git push --help' for details.

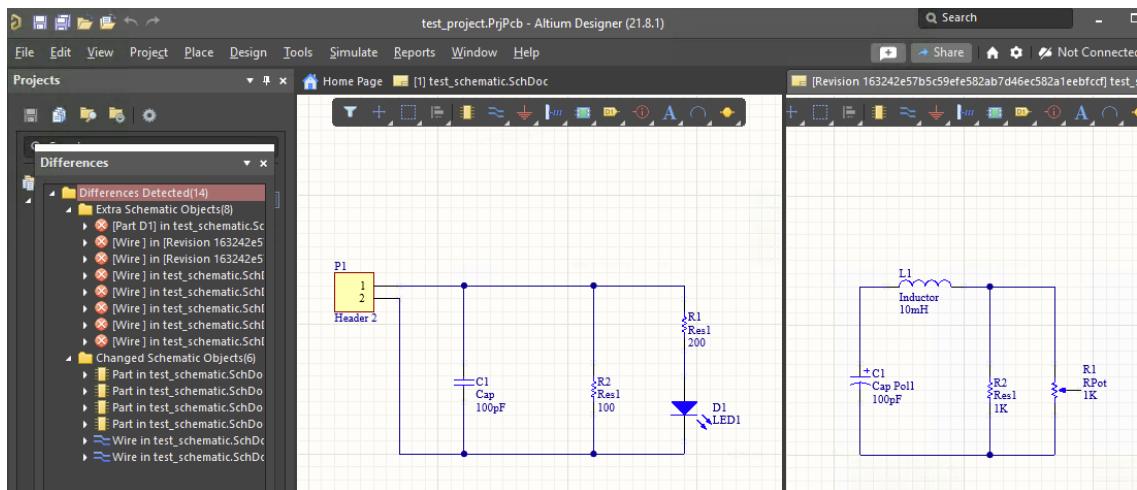
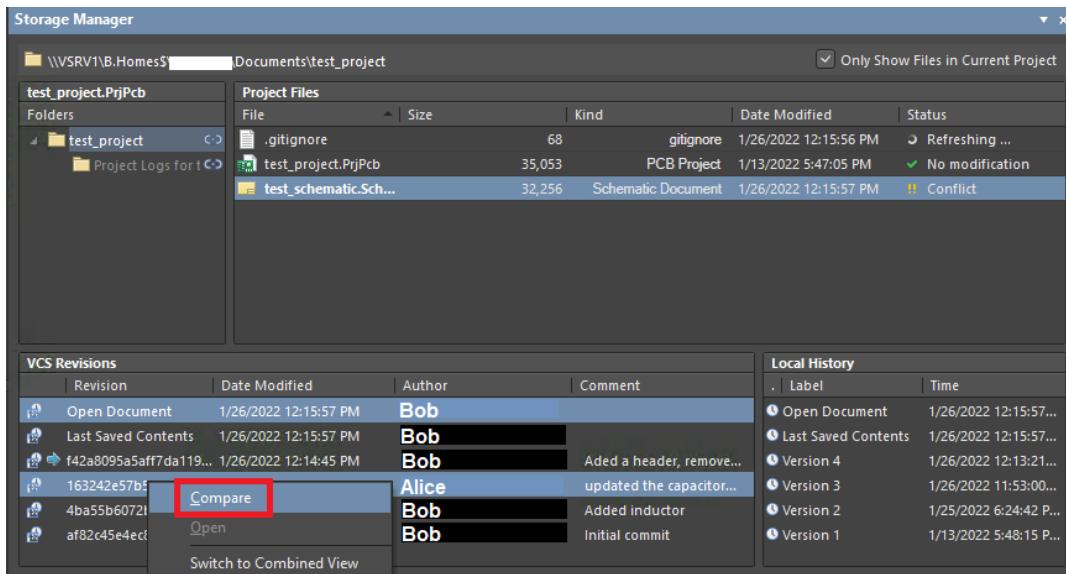
@BA3128WS08 MINGW64 //VSrv1/B.Homes$ cd Documents/test_project (master)
$ git pull
@ug251.eecg.utoronto.ca's password:
remote: Enumerating objects: 7, done.
remote: Counting objects: 100% (7/7), done.
remote: Compressing objects: 100% (4/4), done.
remote: Total 4 (delta 1), reused 0 (delta 0)
Unpacking objects: 100% (4/4), 3.75 KiB | 4.00 KiB/s, done.
From ug251.eecg.utoronto.ca:/nfs/ug/groups/ECE295S/hdc-001/altium_test
 64438b2..163242e master      -> origin/master
warning: Cannot merge binary files: test_schematic.SchDoc (HEAD vs. 163242e57b5c59efe582ab
7d46ec582a1eebfccf)
Auto-merging test_schematic.SchDoc
CONFLICT (content): Merge conflict in test_schematic.SchDoc
Automatic merge failed; fix conflicts and then commit the result.

@BA3128WS08 MINGW64 //VSrv1/B.Homes$ cd Documents/test_project (master|MERGING)
$ |
```

To view the changes Alice made, Bob can use the Storage Manager tool in Altium. To open it, click the **Panels** button at the bottom right of the design workspace then click **Storage Manager**. The Altium guide on this tool is located [here](#).

To resolve the conflict, view both conflicting files, Bob's local copy and the remote commit from Alice by highlighting both files in the **VCS Revisions** table, right-clicking and selecting **Compare**.

This opens both files so the differences can be examined.



In general to resolve non-text file conflicts in git, the two options are to keep the local copy (ours), or keep the remote copy (theirs). In this case Bob may want to incorporate some of Alice's changes and then commit again, thus updating and keeping the local copy.

Once Bob has finished updating his copy, he wants to keep the local files (ours) which can be done with the following commands:

```

1 #Commit the changes Bob made to update the files based on Alice's changes
2 > git status
3 > git add .
4 > git commit -m 'updating files with Alice's work and resolving merge conflict'
5 > git push
6
7 #Or accept the local files without any changes:
8 > git checkout --ours <file_name>
9
10 #To accept the local version for all conflicting files:
11 > git merge --strategy-option ours

```

If instead Bob sees Alice's changes and just wants to accept them and start working again from there he can run

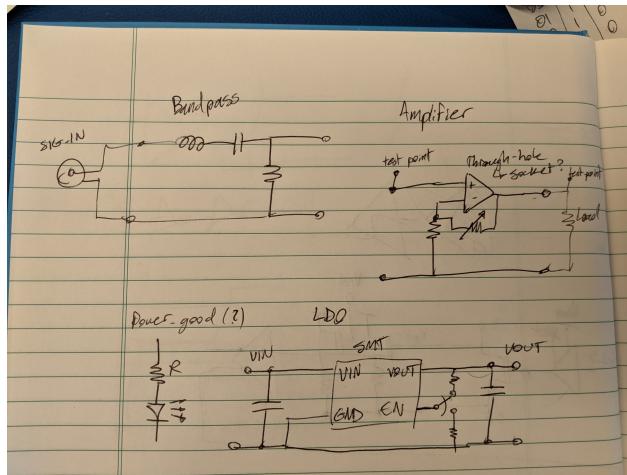
the commands:

```
1 #Accept the remote branch version of a file:  
2 > git checkout --theirs <file_name>  
3  
4 #To accept the local version for all conflicting files:  
5 > git merge --strategy-option theirs
```

:

2 Basic Schematic Capture

Before you bring your schematic to Altium, you should have a basic idea of your design, likely with some sketches that look something like this:

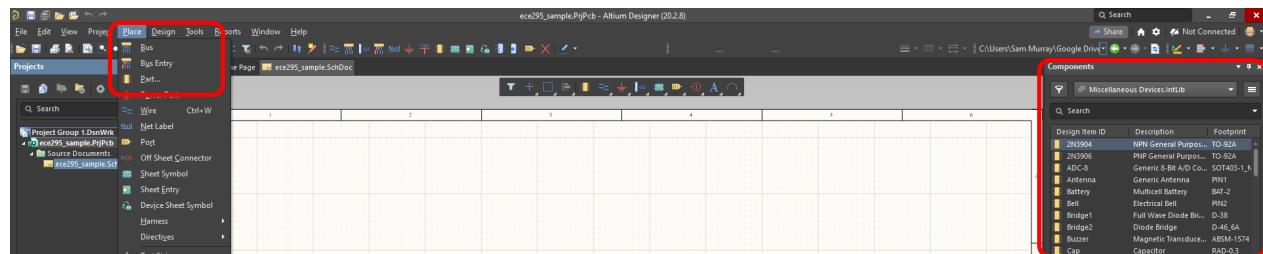


You don't need to have your whole design figured out, but it is helpful to start with some idea of the parts for your design. This lets you start the process of transferring your first ideas to full schematics that can later be made into a PCB.

2.1 Placing Parts

Once your project has a Schematic Document (*.SchDoc file in Altium, see ?? to create this) the first thing to do is to add parts. Go to **Place** → **Part** or you can use the keyboard shortcuts **P** → **P**.

This will bring up the “Components” panel on the right of the screen.



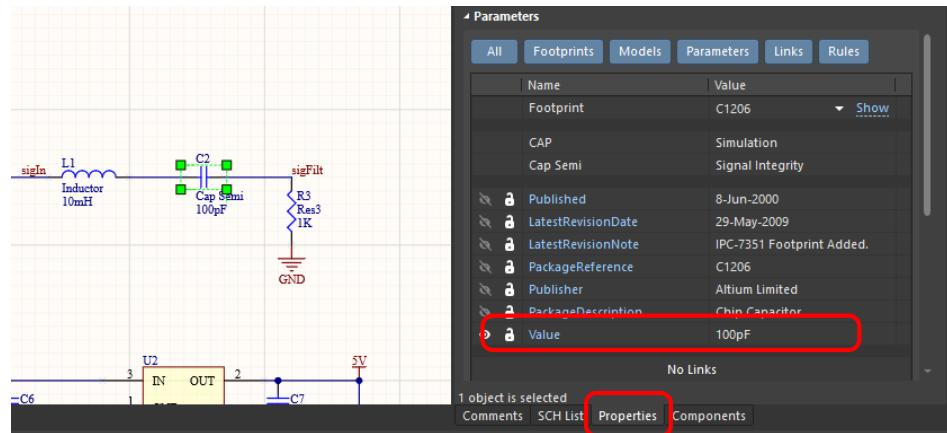
By default, your project will have access to parts from two generic libraries:

- Miscellaneous Devices.IntLib
- Miscellaneous Connectors.IntLib

These parts are the correct choice for if you want to quickly get started and are unsure of your final design parameters they include basic and generic symbols and footprints for parts including:

- | | | | |
|--------------|---------------|------------------|----------------------------|
| • Resistors | • Transistors | • Diodes | • Coaxial cable connectors |
| • Capacitors | • LEDs | • Motors | |
| • Switches | • Relays | • Pin Connectors | |

To use these parts, you can either right click, and select the place option, or click and drag them onto the schematic. Once they are on the schematic you can edit their properties by left-clicking the component and selecting the **Properties** tab on the bottom right. The most important property to update is typically the *Value*, since leaving this as the default can lead to using the incorrect part later on in the design.

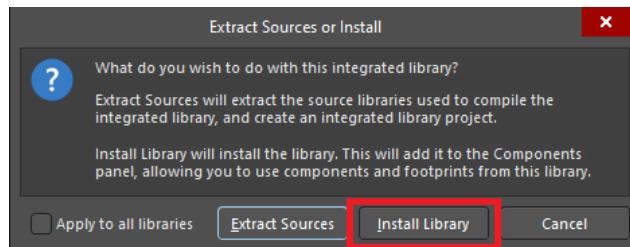


2.1.1 Adding the ECE295 Custom Libraries

Beyond the simple parts available in the two Miscellaneous libraries, there are also a few libraries created specifically for ECE295 to indicate what components will be stocked in the lab by default. These libraries are listed below:

- ECE295-Resistors-TH
- ECE295-Capacitors-TH
- ECE295-Capacitors-SMT
- ECE295-Other

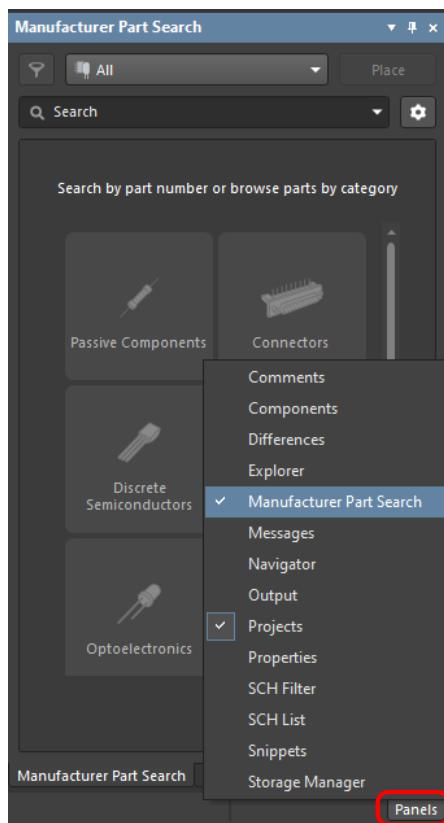
In this case -TH indicates through-hole parts, and -SMT indicates surface mount parts. These libraries are available through *git* here. Either clone or download the .IntLib files and store them somewhere where they will not be accidentally deleted. Then double-click to open and select **Install Library** in the Altium pop-up window that opens. This will locate the ECE295 libraries in the same place as the two Miscellaneous libraries.



Other than the course library, you will likely want to create your own libraries for any specific parts needed in your design. It is also possible to find many useful Altium libraries online. To learn about adding more libraries to your project check out Section 2.6.

2.1.2 Manufacturer Part Search

If you have more information about the part you want to use, you can instead add parts that are contained in Altium's vast Manufacturer Parts section. To access the *Manufacturer Part Search* panel, click the **Panels** button at the lower right corner and select **Manufacturer Part Search**.



An excellent guide on using this feature is provided by Altium here.

It is easiest to use this feature if you already have a part number in mind. It is typically easier to decide on a part by browsing a supplier website. Typical suppliers that export to Canada include:

- Digi-Key <https://www.digikey.ca/>
- Mouser <https://www.mouser.ca/>
- Arrow <https://www.arrow.com/>

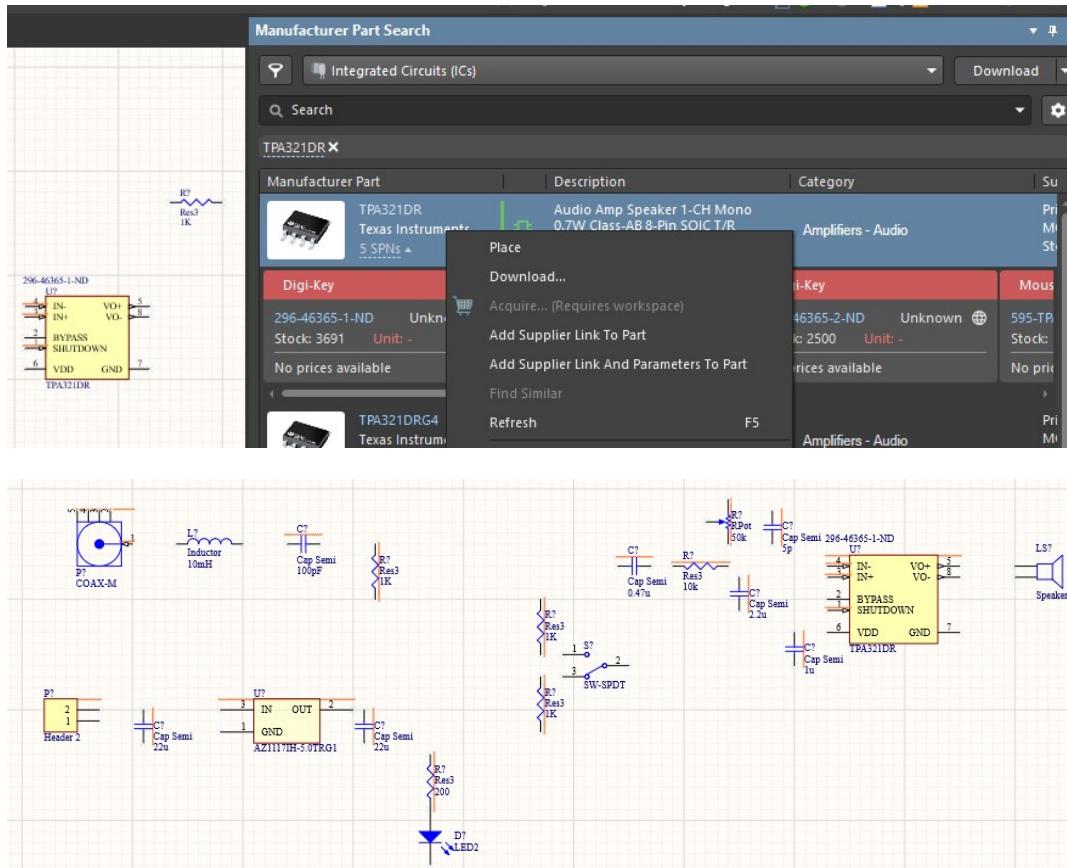
See Appendix A for a guide on searching for components from Digi-Key.

To place a part from the manufacturer search, right click and place, or drag the part to the schematic, the same as adding from the library.

2.2 Example with all parts placed

For the purpose of this guide, our example circuit consists of a bandpass filter, an power conditioning IC (called an LDO, or low dropout regulator), an audio amplifier, a power indication LED, and a speaker to act as the load. All of the selected parts have been placed on the schematic with none of the wiring completed.

This example primarily uses components from the “Miscellaneous” Libraries, except for the ICs, **TPA321DR** and **AZ1117IH-5.0TRG1**. They are from the Manufacturer Part Search, and a self-created symbol in my library (see Section 6.1), respectively.



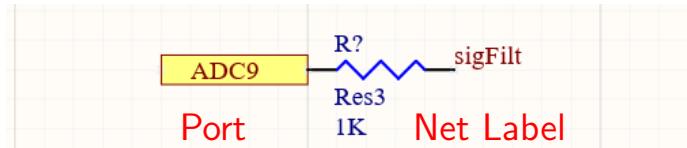
2.3 Wiring, Net Labels, Ports and Power Ports

To start wiring, go to **Place** → **Wire** (or use the shortcut **P** → **W**). Your cursor cross marks will turn blue when you are over top of the pin connection of a component. Left-click to make more turns in the wire, and press **ESCAPE** to exit out of the wire-placing mode.

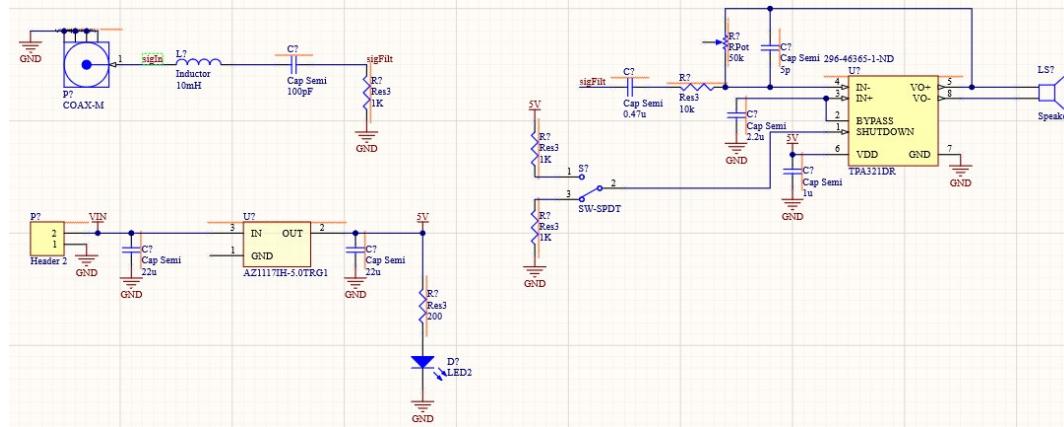
If two wires are crossing each other and there should be a connection, you should see a dot.

Ports, Power Ports and Net Labels let you mark common points on a schematic without wiring things together directly. **Power Ports** should be used for labelling the ground net as well as any power nets present in your schematic. This example has *GND*, *VIN*, and *5V* power ports.

Ports and **Net labels** also connect two different points of the schematic without using a wire.



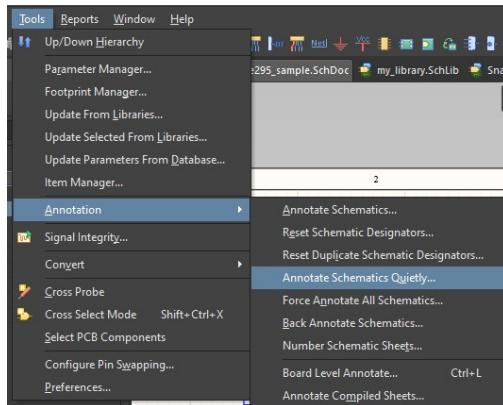
When you place a **net label** onto a connection, it automatically names that net. This is helpful for PCB layout since any unnamed net receives an automatically generated name from Altium that is not descriptive of the net's function. Within a single schematic it is best to use net labels, ports are typically reserved for making connections between different hierarchies. For more on design hierarchies and using ports see Section 6.2. In the example, *sigFilt* joins two points without a wire, and *sigIn* is only a clarifying name.



2.4 Annotation

Once you have wired a schematic and given it sufficient ports and net labels, it is time to annotate. This means replacing all of the question marks in “R?” and “C?” with designated numbers to make identifying components and assembling the PCB later a much easier task.

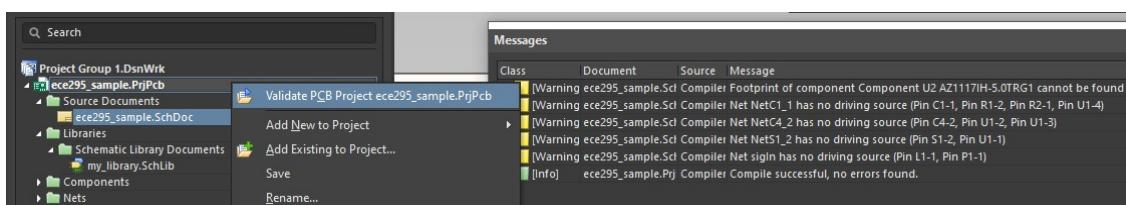
Select **Tools** → **Annotate** → **Annotate Schematics Quietly**. A pop-up will tell you how many designators need to be changed. Click **Yes** and all of the question marks will be updated.



There are variations in how this annotation may be performed, but for our case the default is sufficient.

2.5 Validation

To make sure your schematic is error-free you need to run the validation check. Altium has a pre-defined set of rules that can be screened for by running ERC (Electrical Rule Check). Right-click on the project file (upper left side) and select **Validate PCB Project <project_name>.PrjPCB**.



If you have any errors, the **Messages** panel will open. You should still read the messages even if there are no errors, so to do that select **Panels** → **Messages** on the lower right-hand corner.

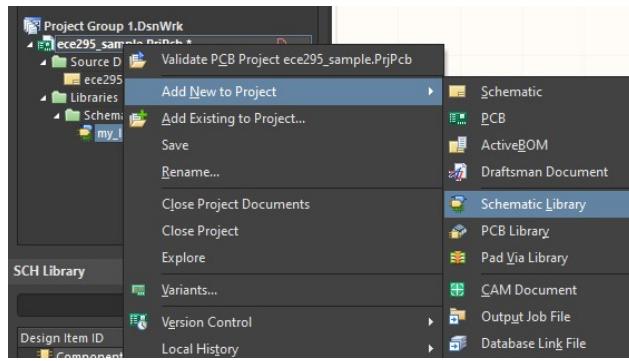
Double-click on the message to zoom to the region of the schematic it is referring to and read carefully to resolve the issue being pointed out. If you are sure you want to suppress an error, you can add a *No ERC* directive, which exists, but this will not be covered in this guide.

2.6 Adding Libraries

Libraries in Altium are repositories for schematic symbols and footprints that you may want to include in your design. As you build your schematic, you can look through the parts in your libraries to be included in the design. A good resource released by Altium to cover this is located here.

There are many different types of libraries in Altium, with the most contained being the .IntLib files used for the course library. The most flexible libraries contain schematics and footprints separately as .SchLib and .PcbLib respectively. It is typically beneficial to have one of each in a project to include any project-specific parts.

You can add a new library to your project by right-clicking the project file in the left Projects pane and selecting **Add New to Project → Schematic Library or PCB Library**



You can also add existing libraries to your project. Similarly, right-click the project file. This time, select **Add Existing to Project** and choose the library files.

3 Pre-PCB Layout Tasks

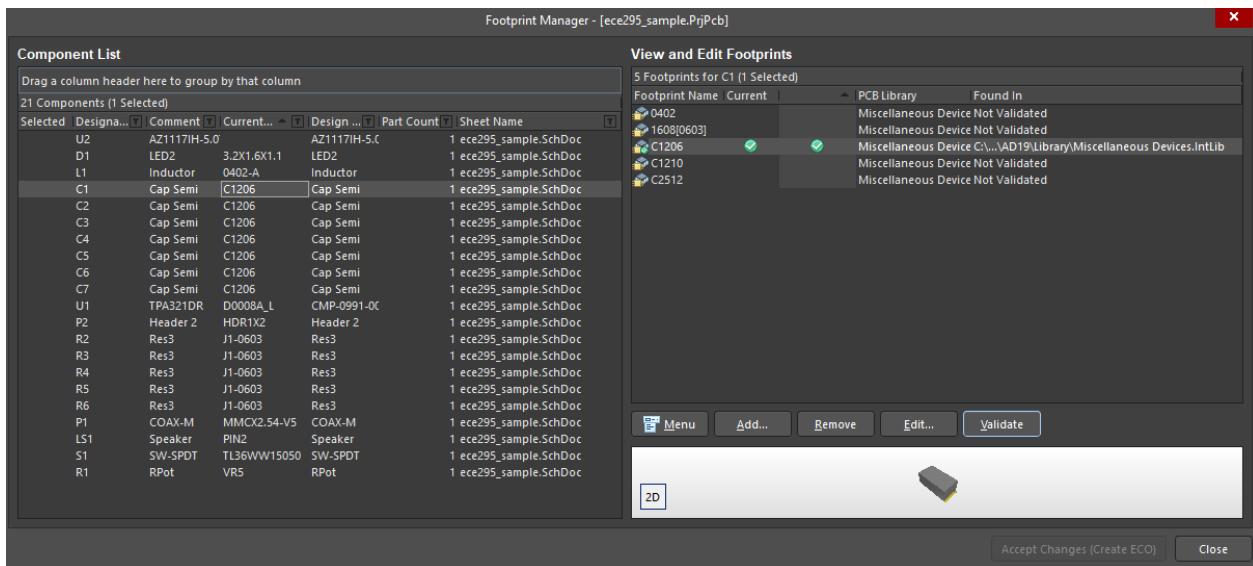
Once your schematic has all components placed, with the appropriate wires and net labels added, it is time to begin preparing for the PCB layout stage. In a schematic each part has pins that correspond to different electrical connections. This must be translated to the physical shape of the component for the PCB. The component shape and electrical connections are described for PCB layout in what is called a part's **footprint**.

Note 2. Prior to generating your PCB and beginning layout it is essential to ensure that each part in your schematic has a PCB footprint, and that the footprint is correct.

Most parts in the *Miscellaneous* libraries already have at least one option for a footprint. Parts from the *Manufacturer Part Search* may have a footprint, (which is actually called a "Model"), but it also may be missing and need to be created before moving to layout. Creating your own footprint will be covered in this section. It is also possible to download footprints for parts from external websites as described in Appendix B.

3.1 Footprint Manager

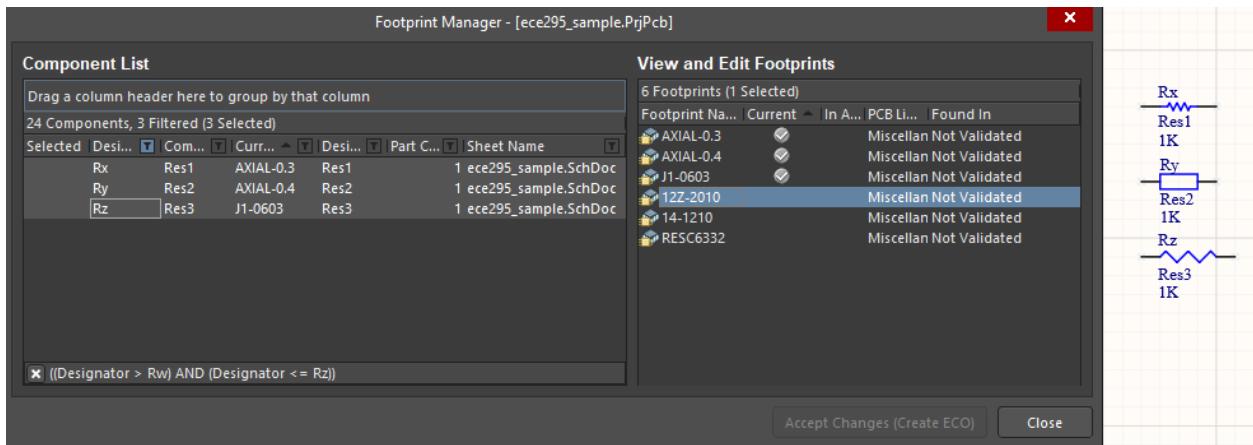
To quickly see what components in your schematic have footprints and what footprint is assigned to each part, open the **Footprint Manager** by selecting **Tools → Footprint Manager**.



Select a part on the left to see its potential footprint options. You can change the selected footprint by right-clicking on the footprint to change to and selecting **Set As Current**.

As an example of changing a default footprint, we can look at the three basic resistor options from the *Miscellaneous Devices* library: **Res1**, **Res2**, **Res3**. Each makes a slightly different shape on the schematic and has different associated footprints. To see a preview of the footprints, select one and make sure the Footprint Manager window is large enough.

To see if any part is missing a footprint, at this stage you need to individually select each component. As this can be tedious for large designs, it is possible to leave this step to when you import your schematic design to the PCB.

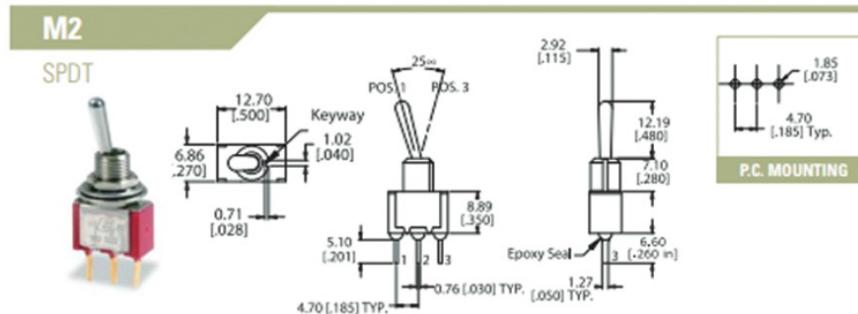


Res1	AXIAL-0.3	Through hole
Res2	AXIAL-0.4	Through hole
Res3	J1-0603	Surface Mount
Res3	12Z-2010	Surface Mount
Res3	14-1210	Surface Mount
Res3	RESC6332	Surface Mount

3.2 Footprint Creation

If there is a missing footprint, then one way to fill in the gap is to create it yourself. In this example we consider the SPDT switch used in the design. By searching on Digi-Key <https://www.digikey.ca/> the part **100SP1T1B1M2REH** (digikey link) is selected.

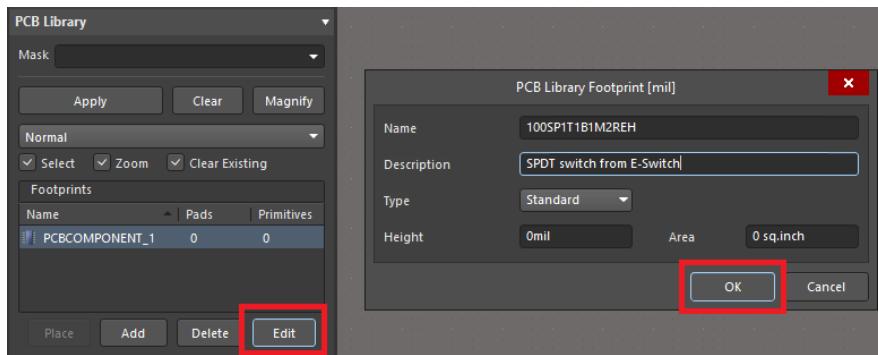
To find its correct footprint, we investigate the datasheet and locate a mechanical description. Here we see the three pins require holes of 1.85mm diameter that are spaced 4.7mm apart.



The dimensions are given in both mm and in inches which is common, although the ordering is not guaranteed, so be sure to check on the datasheet which is listed first. It is also typical to encounter the unit *thou* or *mil* which both refer to one thousandth of an inch.

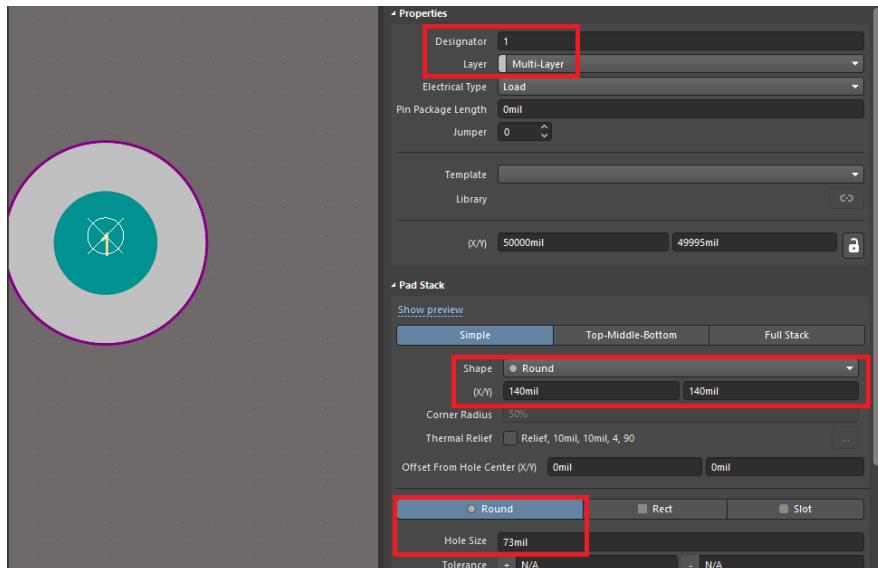
If you do not already have a PCB library, go to **File** → **New** → **Library** → **PCB Library** to create one. By default the newly created library will have one component. Select **Edit** and then put the part number in for the component name. Next we add the three required holes at the correct spacing. Select **Place** → **Pad** (or use the shortcut **P** → **P**). While hovering with the pad you can press **TAB** to open the **Properties** menu and edit the pad.

To create a hole through the PCB your pad type should be set to *Multi-Layer*. This part has three pads, and

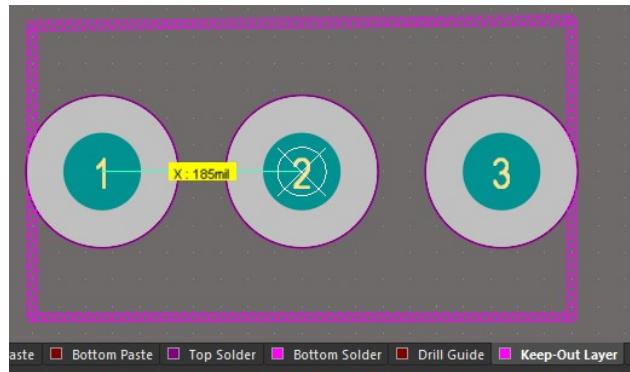


so they should be named with designators 1, 2, and 3 which should also correspond to pins 1, 2, and 3 on the schematic.

The *Hole Size* should match to the value given in the datasheet as P.C. MOUNTING. Here it is set to 73mil, corresponding to 0.073 inches. Remember in Altium to change between mm and mil you can press Q. The value under *Pad Stack* controls how much copper is around the hole for soldering. A good rule of thumb is to set the *Pad Stack* diameter to approximately twice the hole width. It should never be smaller than +20mil from the hole size, and anything beyond +200mil is likely excessive.

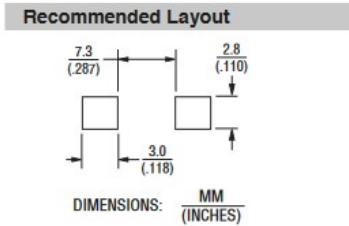


After placing the three holes in the correct location with the correct designators, the footprint is technically complete. You can optionally add an outline of the physical shape of the component on either the silkscreen layer (Top Overlay) or another mechanical layer, such as the Keep-Out Layer. To help in aligning the pads, make use of Altium's coordinate system, and the **Ctrl+M** measurement tool.



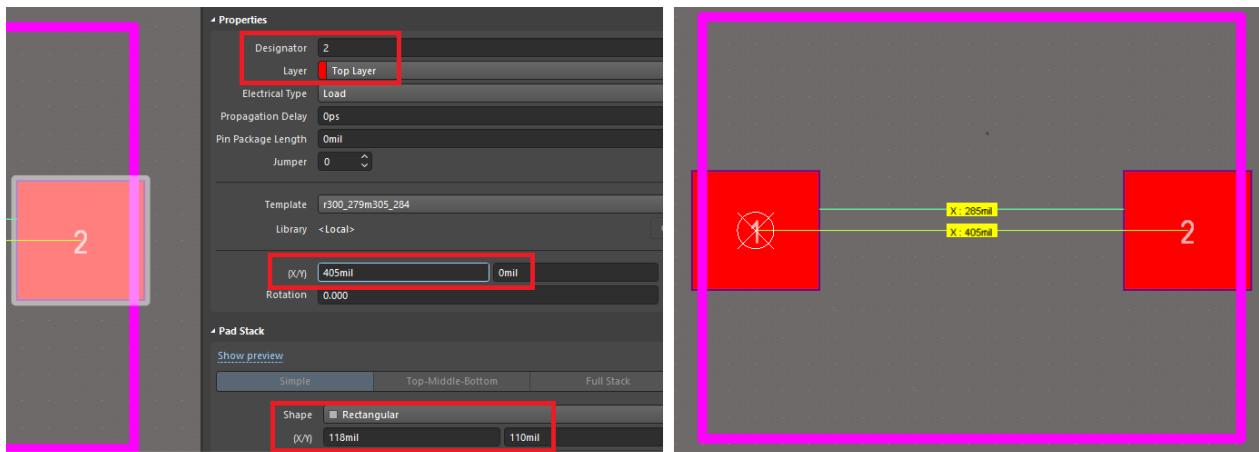
For a second PCB footprint creation tutorial, we will create a footprint for the 10 mH inductor. The selected part is

SDR1005-103KL (digikey link) with a recommended layout provided on the datasheet.



In the PCB library under the footprints list, select Add and then Edit and input the part number as the footprint name. This inductor is a *Surface Mount* part, meaning that it attaches to only one side of the PCB and does not require holes drilled through the board.

Because of this after we select **Place** → **Pad** and hit **TAB** the *Layer* field should be changed to *Top Layer*. Under *Pad Stack*, the shape should also be changed to rectangular and the dimensions from the datasheet can be entered. Under the properties menu it is also possible to precisely change the position of a pad, which can be helpful when aligning objects to match the correct dimensions.

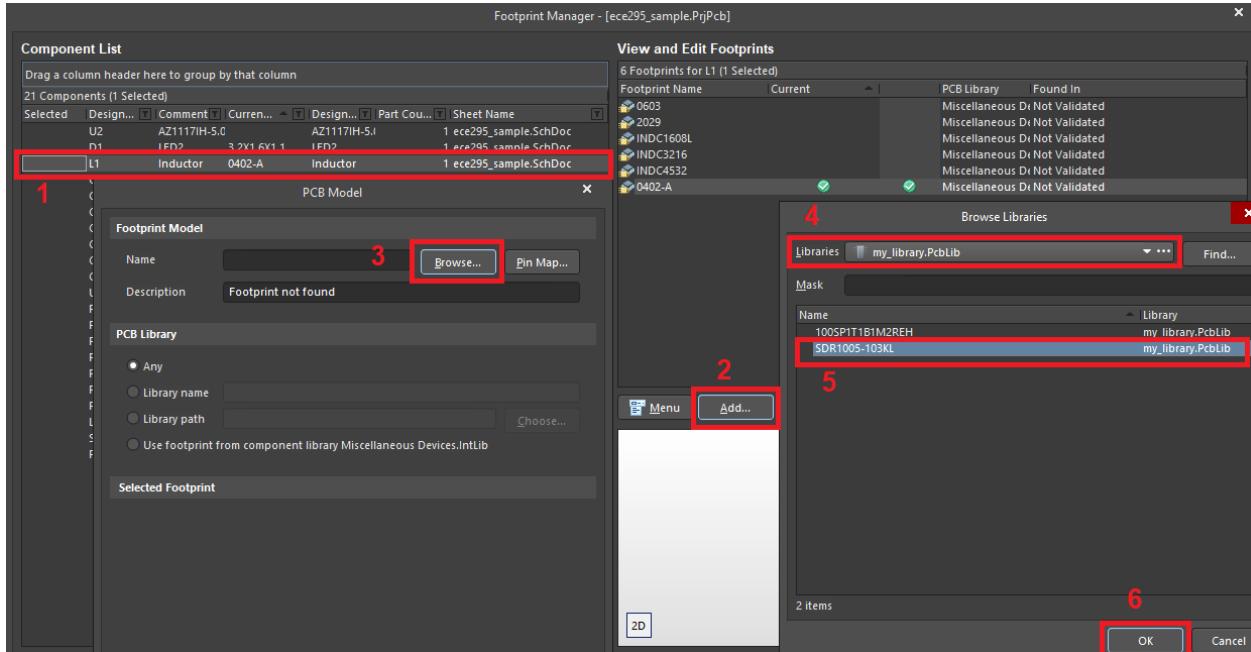


To link the newly created footprints to the relevant parts, first ensure the library is part of the project. You can do this by dragging it into the project on the left sidebar or by right-clicking and selecting add to project.

Then go to **Tools** → **Footprint Manager**, select the appropriate part, click to **Add** a new footprint, **Browse** to

the correct library and component, then click **OK**. Repeat this process for all footprints that you need to add or change.

If the green check mark does not appear under **Current** for newly added footprint, right-click that footprint and select **Set As Current**



3.3 Linking Part Numbers

Either before, during, or after the footprints for each item have been sorted out, it is important to assign, or link, a specific part number to each part.

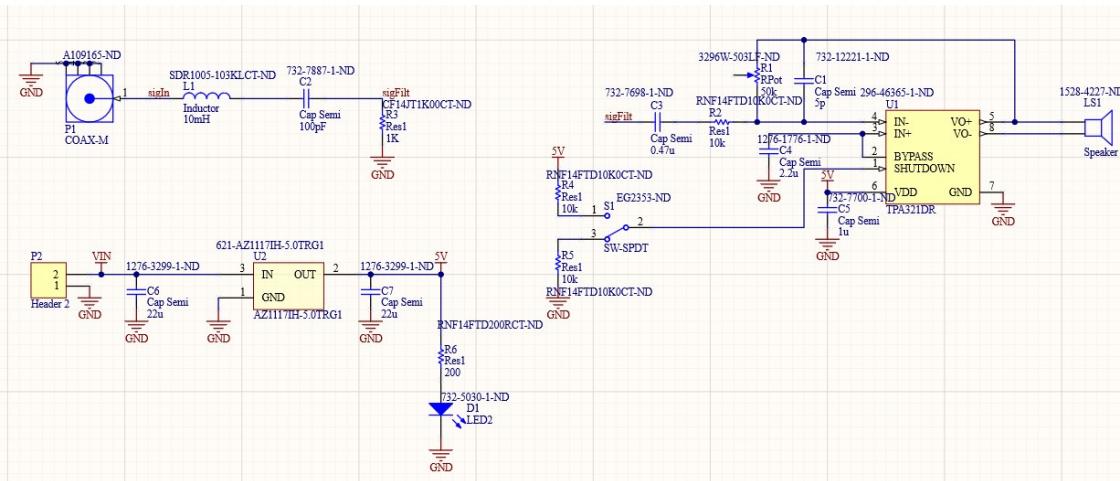
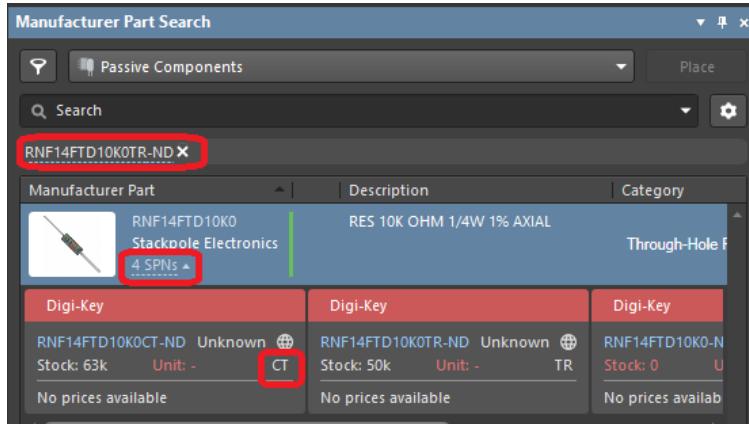
This helps when creating the *Bill of Materials (BOM)* that will be used to order all of the parts required to build the PCB. To link parts, the easiest way is often to use the **Manufacturer Part Search** accessed under the **Panels** button at the lower right corner. If you have placed parts using **Manufacturer Part Search** they are already linked.

To link a part, if you already have a part number, enter it into the search bar, locate the correct component. This process goes the smoothest if you have already have a supplier part number (SPN) for each part from Digi-Key, Mouser, or Arrow. Select your part following the guidelines in Appendix A. Click down where it says **X SPNs**, where X will be a number.

When multiple **SPNs** are available this refers to the same part in a variety of different packaging options from the manufacturer. In this example the highlighted *CT* represents "Cut Tape" and the *TR* in the second box refers to "Tape & Reel". For small designs it is important to select "Cut Tape" as "Tape & Reel" parts will result in being sent a large plastic reel along with your parts. These reels are intended for use with *Pick-and-place* machines that can automatically put components onto PCBs with additional setup.

Once you have selected a supplier block, right click and select **Add Supplier Link And Parameters To Part**. A grey cross will appear and you click on each relevant part to link it. Press **Esc** when done.

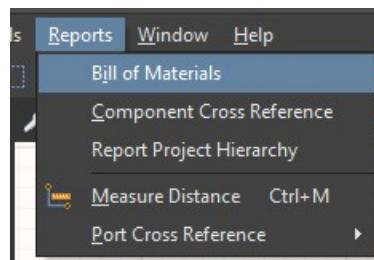
Complete the part linking for all components in the design. The final product should have supplier part numbers for each item. A part can be left unlinked if it is coming from the stock of generic components supplied in the lab.



3.4 BOM Generation and Validation

In order to make your PCB it will at some point be important to acquire the components to be soldered onto the board. A BOM, or *Bill of Materials* is used to designate a specific part to purchase for each item on the PCB. They are typically presented in a table, and link part numbers to designators on the PCB.

Altium has a tool to help generate a BOM that can be found under **Reports → Bill of Materials**



Once the tool is open it is important to add the part numbers. Either the **Manufacturer Part Number** or the **Supplier Part Number** may be used, but it is the **Supplier Part Number** that will ultimately be used to purchase the parts. If the components have already been linked, this field will already be filled.

To add the **Supplier Part Number** field to the BOM, go to the **Columns** tab, then search for the column. Once you have found it, click the eye icon to make it visible in the table and then drag it into the grouping area so that

your table is sorted by **Supplier Part Number**.

Bill of Materials for Project [ece295_sample.PjPcb] (No PCB Document Selected)

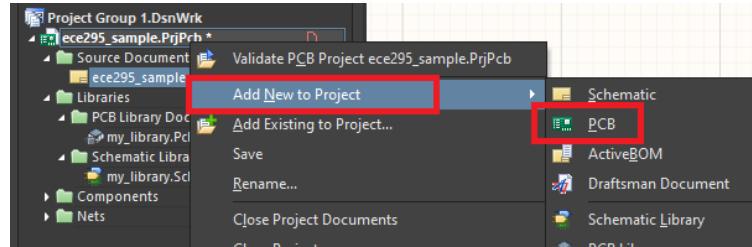
Comment	Description	Designator	Footprint	LibRef	Quantity	Supplier Part N...
1	Cap Semi	C1	C1206	Cap Semi	1	732-12221-1...
2	Cap Semi	C2	C1206	Cap Semi	1	732-7887-1-ND
3	Cap Semi	C3	C1206	Cap Semi	1	732-7698-1-ND
4	Cap Semi	C4	C1206	Cap Semi	1	1276-1776-1...
5	Cap Semi	C5	C1206	Cap Semi	1	732-7700-1-ND
6	Cap Semi	C6, C7	C1206	Cap Semi	2	1276-3299-1...
7	LED2	D1	3.2X1.6X1.1	LED2	1	732-5030-1-ND
8	Inductor	L1	SDR1005-103KL	Inductor	1	SDR1005-103...
9	Speaker	L51	PIN2	Speaker	1	1528-4227-ND
10	COAX-M	P1	MMCX2.54-V5	COAX-M	1	A109165-ND
11	Header 2	P2	HDR1X2	Header 2	1	
12	RPot	R1	VRS	RPot	1	3296W-503LF...
13	Res1	R2, R4, R5	AXIAL-0.3	Res1	3	RNF14FTD10K...
14	Res1	R3	AXIAL-0.3	Res1	1	CF14IT1K00C...
15	Res1	R6	AXIAL-0.3	Res1	1	RNF14FTD200...
16	SW-SPDT	S1	100SP1T1B1M2REH	SW-SPDT	1	EG2353-ND
17	TPA321DR	U1	D0008A_L	CMP-0991-00044-2	1	296-46365-1...
18	AZ1117IH-5.0TRG1	U2		AZ1117IH-5.0TRG1	1	621-AZ1117IH...

This table can then be exported to a .xlsx file that may be uploaded to a part supplier's website to easily purchase all of the required components.

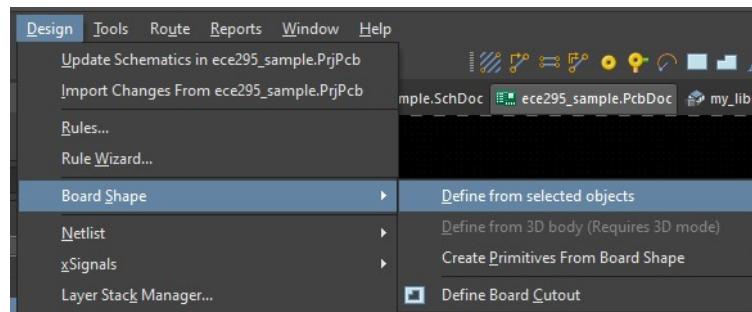
4 Creating Your PCB

4.1 Add a PCB Document and Define a Board Shape

To add a PCB to your project, simply right-click on the project file and select **Add New to Project → PCB**. This will generate a new blank PCB. The first time you save it will be given the opportunity to rename it.



To change the shape of the PCB, the easiest method is to draw the desired board outline using one of the layers, select the outline and navigate to **Design → Board Shape → Define from selected objects**. To draw the lines defining the outline, select **Place → Line** or use shortcut **P → L**. The board outline should be defined on a layer that is not used for anything else in the design. In Altium, it is best to use either the Keep-Out Layer or an otherwise unused Mechanical Layer for the board outline.

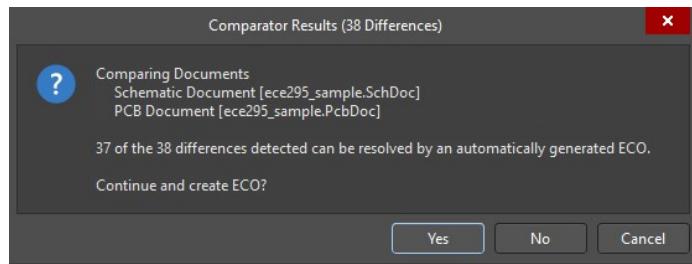


It is also possible to change the location of the coordinate system origin by selecting **Edit → Origin → Set**



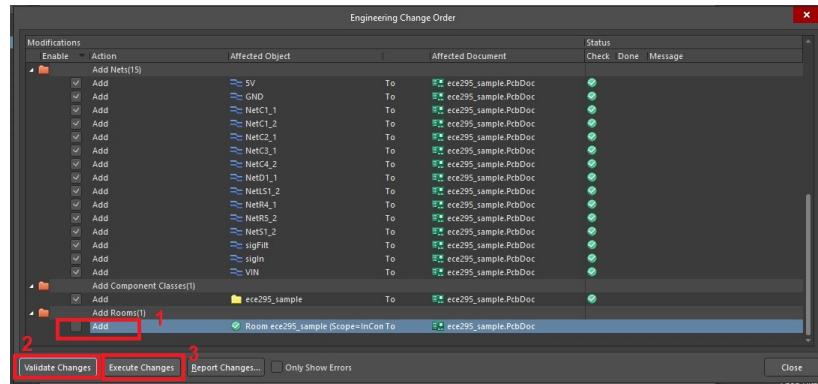
4.2 Synchronizing with Schematic and Importing

Next, the parts from schematic must be brought onto the PCB to be placed and connected. Select **Design → Import Changes From <your_project_name>.PrjPcb**. A pop-up will appear, click **Yes**.

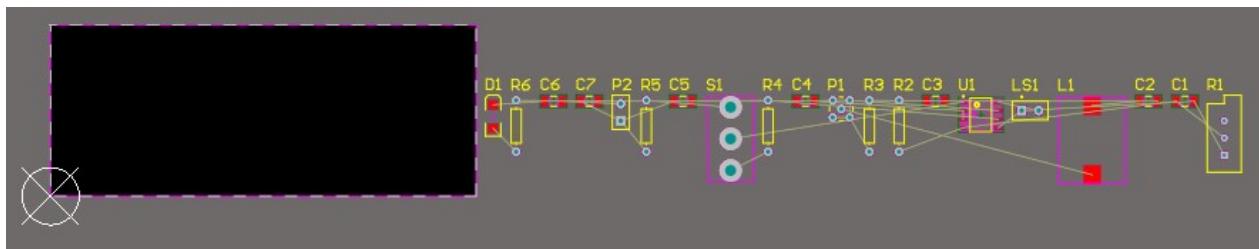


A new window will open listing the actions to be taken to update the PCB to reflect the schematic. In general, it is good to allow all of these things to happen, although it is advisable to un-check the item listed under **Add Rooms** as a beginner to avoid some complications.

Next select **Validate Changes** then **Execute Changes** if green check marks appear under the *Status Check* column.



After execution, close the Engineering Change Order window and the footprints of each part on the schematic should have appeared beside the PCB with thin grey lines connecting them. These lines are called the “rat’s nest” and they represent the connectivity defined between parts in the schematic. As the parts are placed on the PCB and connected with copper lines on either the top or bottom layer, the rat’s nest lines disappear.

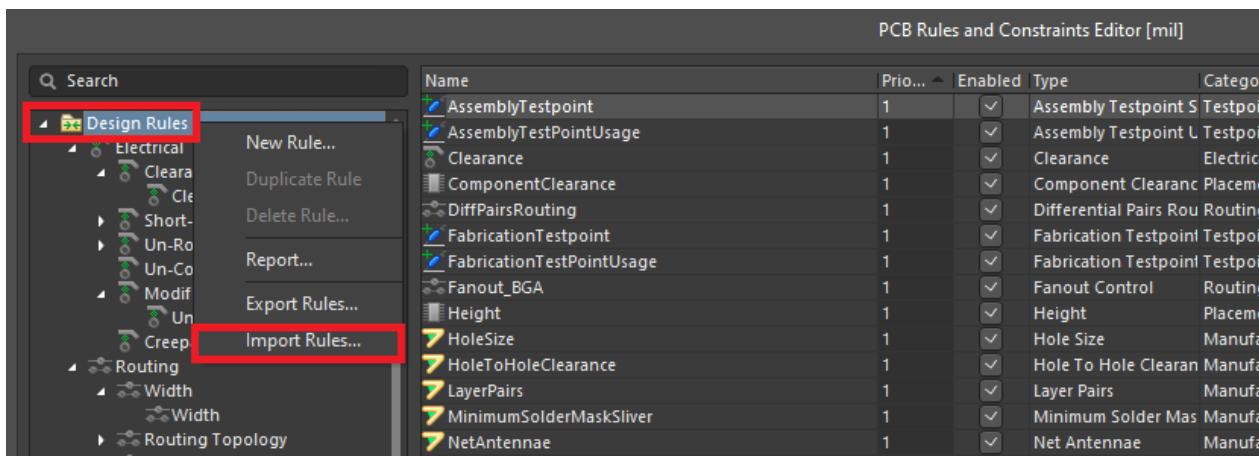


4.3 Configuring PCB Manufacturing Rules

Before beginning to place parts and route wiring between them, it is important to set up the manufacturing rules. This ensures that Altium can provide an error if during routing parts are placed too close together, wires are too thin, holes are too small, or any number of other rules are violated.

To access the PCB design rules from the top ribbon select **Design** → **Rules...**. Here all of the default rules may be changed.

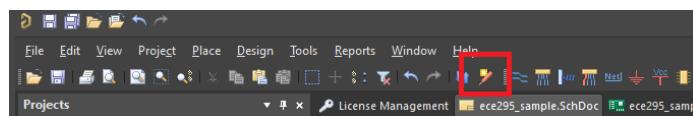
For the purpose of this course, since all PCBs will be manufactured to the same standards, a rule file will be provided for you to upload. Scroll up and right-click on the **Design Rules** folder, then select **Import Rules...**



This opens the **Choose Design Rule Type** panel. Select all of the rule types using **Ctrl+A** and hit **OK**. Once the option to load a **.rul** file appears, select the provided course file.

4.4 Layout Guidelines

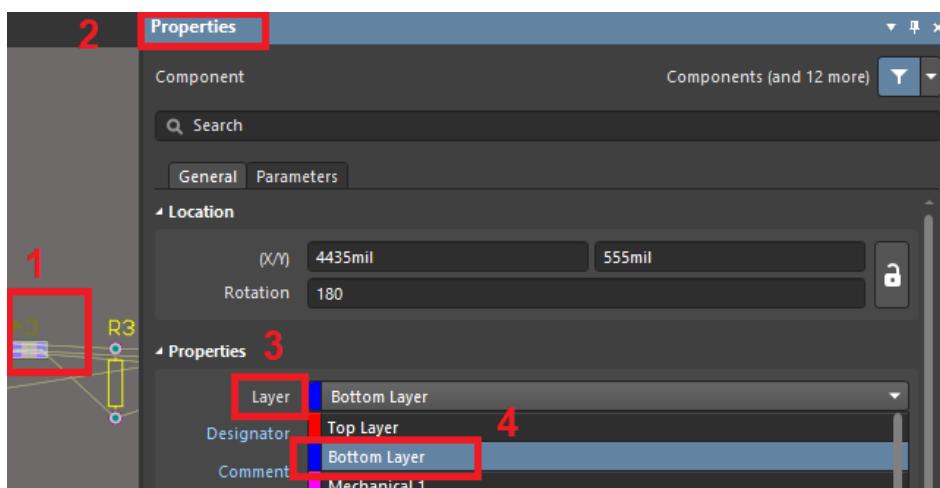
To start layout, the first step is placing the components onto the PCB shape in an order that makes sense to how they are connected in the schematic. For this stage it may be helpful to pull either the schematic or PCB tab out of the main Altium window so you can view both at once. Do this by dragging one of the tabs outside the Altium window. Another helpful feature is the *Cross Probe* tool. By selecting this tool from the ribbon, once a net is selected in the PCB or schematic, the same net is highlighted in the opposite view. To clear a highlight, press **Shift+C**.



To highlight an individual net in the PCB view to see all of its connections, press **Ctrl+C**.

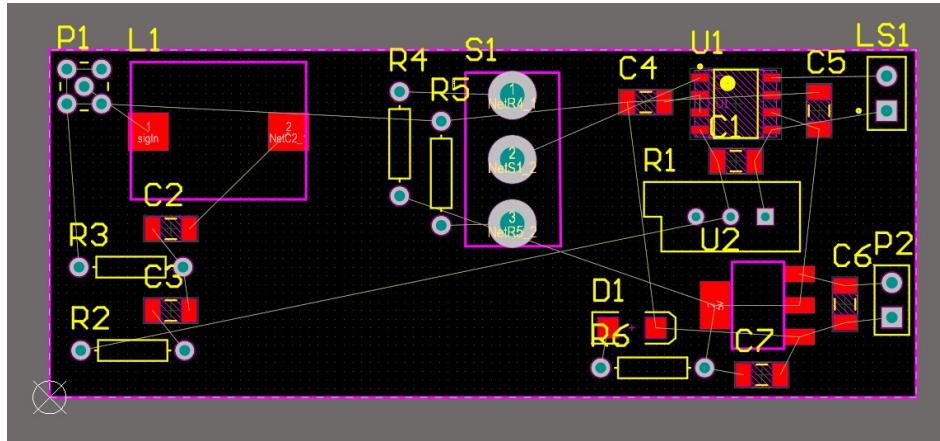
Parts may be dragged and while holding a part or with a part selected, press **SPACE** to rotate.

To place a component on the bottom of the PCB, select it, open the Properties Panel (right-click Properties... or Panels → Properties from the bottom right corner)



After this the component designator (C4 in this example) should become inverted. If the component is surface mount, the pad colour should change to match the bottom layer colour.

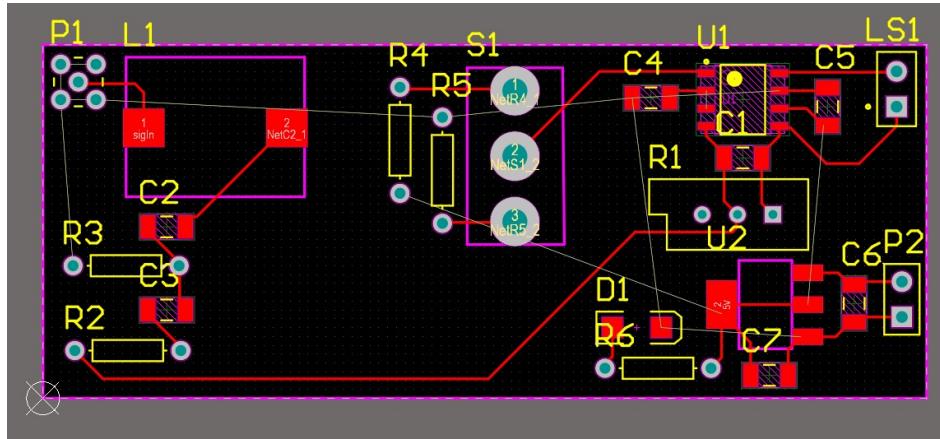
After placing all of the components all parts should be on the PCB. At this stage let the yellow designators overlap onto other components, they can be moved once routing is complete.



To route, select the layer to begin on and select **Place** → **Track** (or use the shortcut **P** → **T**). By default routing will bend only at 45 degree angles. To cycle through routing options, press **SPACE** while routing. It is good to stick with the default in most cases.

It is also possible to drag tracks that have already been placed to move them around.

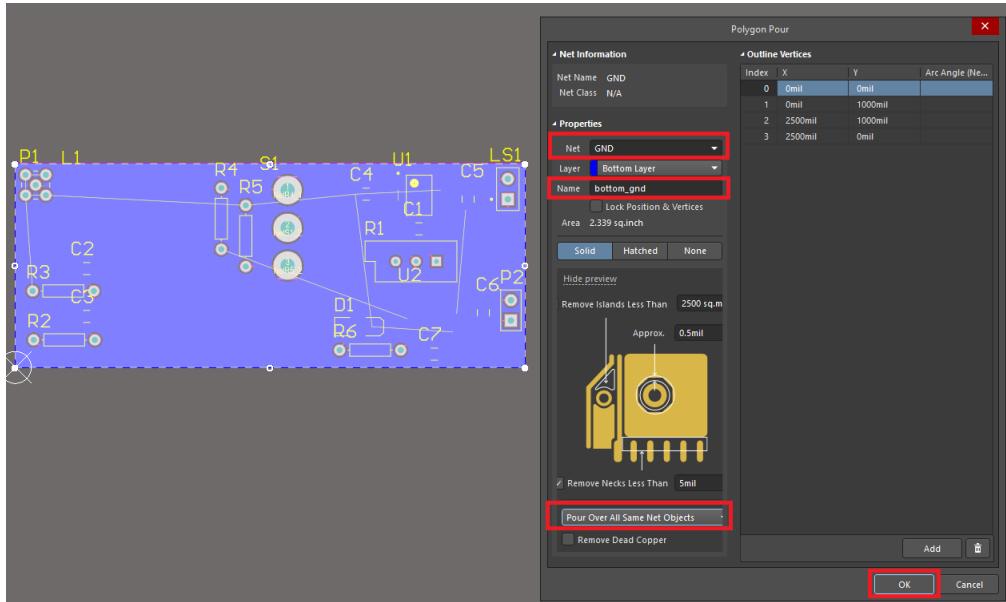
When routing a PCB it is good practice to start with the signal traces, and leave the power ports to the end. In this example everything except the 5V and GND nets have been routed.



To ensure good connection and higher current-handling capability, it is often desirable to route the power rails as large rectangular planes of copper rather than as thin lines. To do this in Altium, the **Polygon** feature is used.

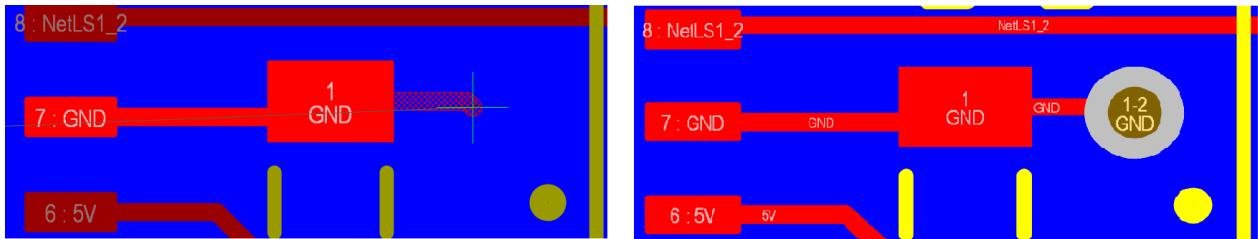
Select **Place** → **Polygon** (shortcut **P** → **G**) and route the edges that will define where copper will be poured. In the example below, the entire bottom of the board is filled with copper to be connected to the **GND** net. Once the polygon is drawn, the **Polygon Pour** window opens. Here, select the net to connect the Polygon too, the layer it should be poured on, and give it a name.

It is also good to change the menu at the bottom to **Pour Over All Same Net Objects** so the polygon does not go around any traces you may have already routed on the GND net.



If the polygon ever goes green with errors, select it, right-click and go to **Polygon Actions** → **Repour Selected**. Polygons can also be managed under **Tools** → **Polygon Pours** → **Polygon Manager**.

Vias are used to get from one layer to another for routing or connection purposes. The easiest way to switch layers while routing (**P** → **T**) is to hold **Shift+Ctrl** and use the scroll wheel to change layers. To move to a new layer, a via will automatically appear. Simply click to place it and hit **Esc** to finish routing.



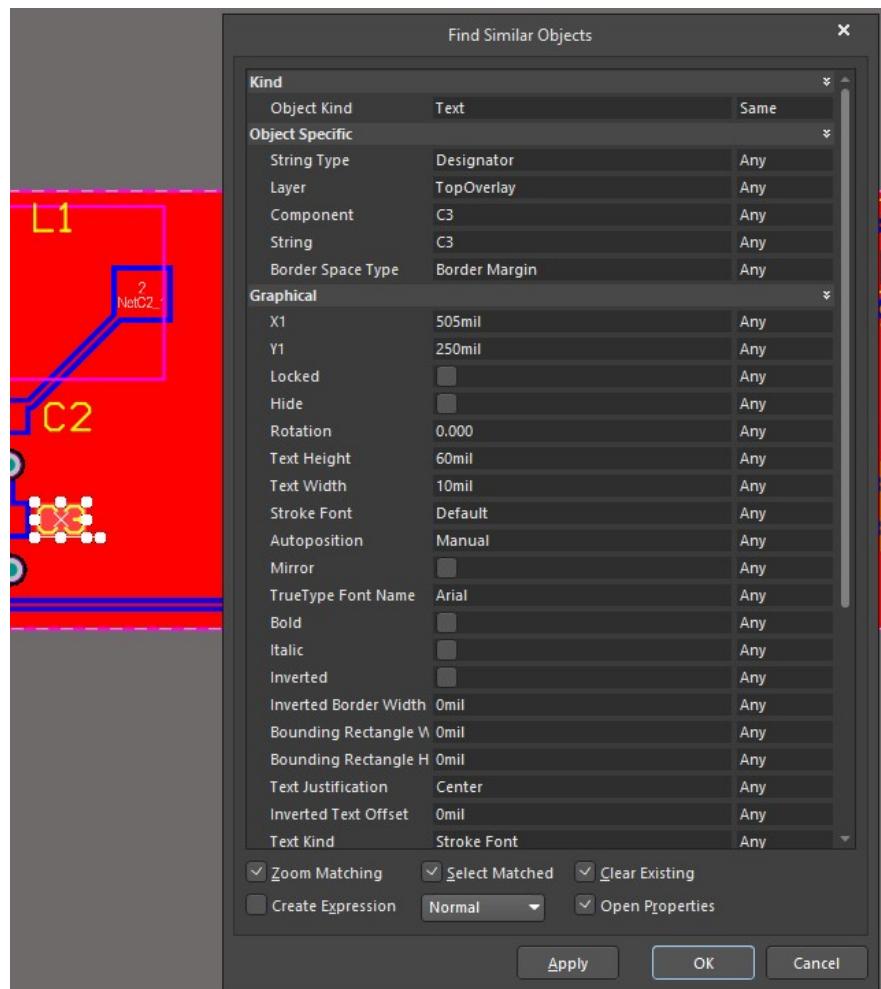
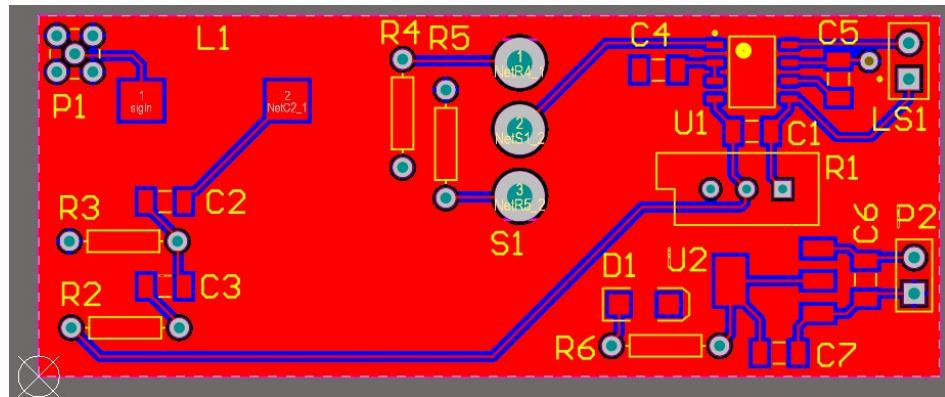
Vias may also be placed using **Place** → **Via** (shortcut **P** → **V**). Pressing **TAB** while hovering over the via brings up the **Properties** menu where the size and net connectivity of the via can be changed.

Once all nets are routed, it is important to go to the **Top Overlay** layer and move all of the component designators so that they are on the PCB and not overlapping with other components. This provides a manufacturing aid; when assembling the PCB it is easy to uniquely identify each component by number.

Occasionally when routing it is desirable to edit many of the same thing at once. Altium's tool for this is called **Find Similar Objects**. To access the menu, select an item, right-click, and select **Find Similar Objects**.

Change the items in the search field and the type to select between *Any*, *Same*, and *Different*. Select **Apply** to select all of the identified objects.

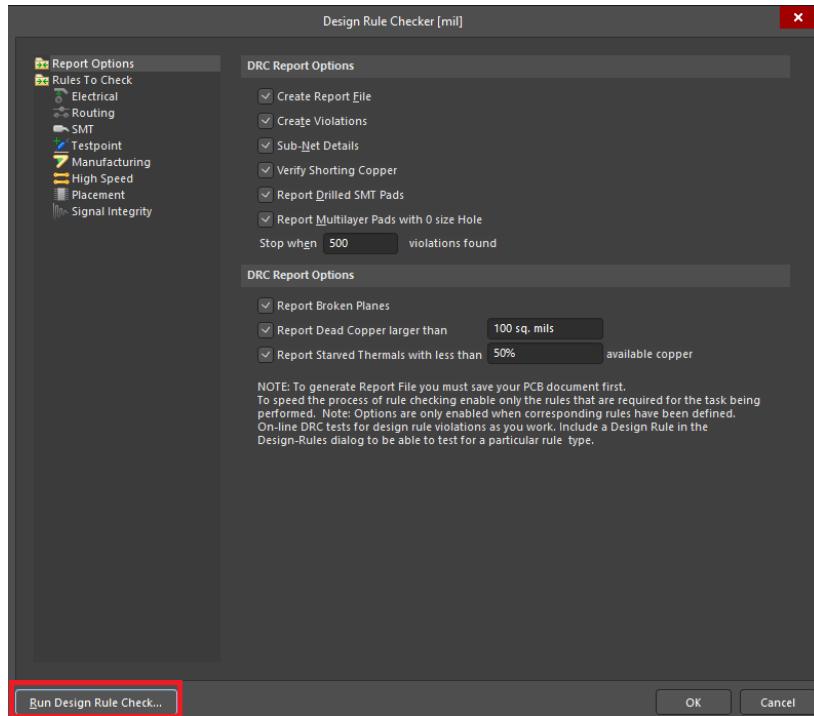
Once the objects are selected and highlighted, select **OK** to close the menu. Now editing fields under the **Properties** menu edits that property for all selected items.



4.5 Design Rule Check

Once the PCB routing is complete or almost complete, it is important to run a design rule check to ensure that all rules have been followed and your design has been correctly copied from the schematic to the PCB.

To open the Design Rule Check, go to **Tools → Design Rule Check...** and a window will open. In most cases, simply press **Run Design Rule Check....**



Running the Design Rule Check (DRC) will open two windows: the **Altium Messages** panel, and a new window called the **Design Rule Verification Report**. The report summarizes all of the errors and sorts them into categories. To find a rule violation in the PCB, click either the link in the report or messages panel to be snapped to the violation location in the PCB.

Class	Document	Source	Message	Time	Date	No.
[Un-Routed N]	ece295_sample.PcbDoc	Advanced F	Un-Routed Net Constraint: Net GND Between Pad C4-1(1666.85mil,850mil) on Top Layer And Pad U1-7 13:07:31	2021-10-25	1	
[Un-Routed N]	ece295_sample.PcbDoc	Advanced F	Un-Routed Net Constraint: Net GND Between Pad D1-2(1748.89mil,200mil) on Top Layer And Track (2 13:07:31	2021-10-25	2	
[Un-Routed N]	ece295_sample.PcbDoc	Advanced F	Un-Routed Net Constraint: Net 5V Between Pad R4-2(1010mil,580mil) on Multi-Layer And Track (1920m 13:07:31	2021-10-25	3	
[Un-Routed N]	ece295_sample.PcbDoc	Advanced F	Un-Routed Net Constraint: Net 5V Between Pad U2-2(2164mil,265mil) on Top Layer And Track (2208.18 13:07:31	2021-10-25	4	
[Minimum Sol]	ece295_sample.PcbDoc	Advanced F	Minimum Solder Mask Sliver Constraint: (7.593mil < 10mil) Between Pad P1-1(100mil,895mil) on Multi-L 13:07:31	2021-10-25	5	
[Minimum Sol]	ece295_sample.PcbDoc	Advanced F	Minimum Solder Mask Sliver Constraint: (7.593mil < 10mil) Between Pad P1-1(100mil,895mil) on Multi-L 13:07:31	2021-10-25	6	
[Minimum Sol]	ece295_sample.PcbDoc	Advanced F	Minimum Solder Mask Sliver Constraint: (7.593mil < 10mil) Between Pad P1-1(100mil,895mil) on Multi-L 13:07:31	2021-10-25	7	
[Minimum Sol]	ece295_sample.PcbDoc	Advanced F	Minimum Solder Mask Sliver Constraint: (7.593mil < 10mil) Between Pad P1-1(100mil,895mil) on Multi-L 13:07:31	2021-10-25	8	
[Silk To Solder]	ece295_sample.PcbDoc	Advanced F	Silk To Solder Mask Clearance Constraint: (9.692mil < 10mil) Between Pad R2-1(90mil,135mil) on Multi- 13:07:31	2021-10-25	9	
[Silk To Solder]	ece295_sample.PcbDoc	Advanced F	Silk To Solder Mask Clearance Constraint: (9.787mil < 10mil) Between Pad R2-2(90mil,135mil) on Mult 13:07:31	2021-10-25	10	
[Silk To Solder]	ece295_sample.PcbDoc	Advanced F	Silk To Solder Mask Clearance Constraint: (9.692mil < 10mil) Between Pad R3-1(95mil,375mil) on Multi- 13:07:31	2021-10-25	11	
[Silk To Solder]	ece295_sample.PcbDoc	Advanced F	Silk To Solder Mask Clearance Constraint: (9.787mil < 10mil) Between Pad R3-2(385mil,375mil) on Mult 13:07:31	2021-10-25	12	
[Silk To Solder]	ece295_sample.PcbDoc	Advanced F	Silk To Solder Mask Clearance Constraint: (9.644mil < 10mil) Between Pad R4-1(1010mil,880mil) on Mu 13:07:31	2021-10-25	13	
[Silk To Solder]	ece295_sample.PcbDoc	Advanced F	Silk To Solder Mask Clearance Constraint: (9.644mil < 10mil) Between Pad R4-2(1010mil,580mil) on Mu 13:07:31	2021-10-25	14	
[Silk To Solder]	ece295_sample.PcbDoc	Advanced F	Silk To Solder Mask Clearance Constraint: (9.644mil < 10mil) Between Pad R5-1(1130mil,795mil) on Mu 13:07:31	2021-10-25	15	
[Silk To Solder]	ece295_sample.PcbDoc	Advanced F	Silk To Solder Mask Clearance Constraint: (9.644mil < 10mil) Between Pad R5-2(1130mil,495mil) on Mu 13:07:31	2021-10-25	16	
[Silk To Solder]	ece295_sample.PcbDoc	Advanced F	Silk To Solder Mask Clearance Constraint: (9.692mil < 10mil) Between Pad R6-1(1590mil,85mil) on Mult 13:07:31	2021-10-25	17	
[Silk To Solder]	ece295_sample.PcbDoc	Advanced F	Silk To Solder Mask Clearance Constraint: (9.787mil < 10mil) Between Pad R6-2(1890mil,85mil) on Mult 13:07:31	2021-10-25	18	

Resolve the errors by addressing each of them individually and continue running the DRC until no rule violations are reported. Occasionally it is necessary to leave a violation in a design. Please check with the TAs before doing that for the purposes of this course.

The screenshot shows the Altium Designer interface with the title "Design Rule Verification Report". The report summary indicates 0 Warnings and 18 Rule Violations. The "Summary" section provides a detailed breakdown of these violations across various categories.

Category	Count
Warnings	0
Total	0
Rule Violations	18
Clearance Constraint (Gap=10mil). (All).(All)	0
Short-Circuit Constraint (Allowed=No). (All).(All)	0
Un-Routed Net Constraint. (All)	4
Modified Polygon (Allow modified: No), (Allow shelved: No)	0

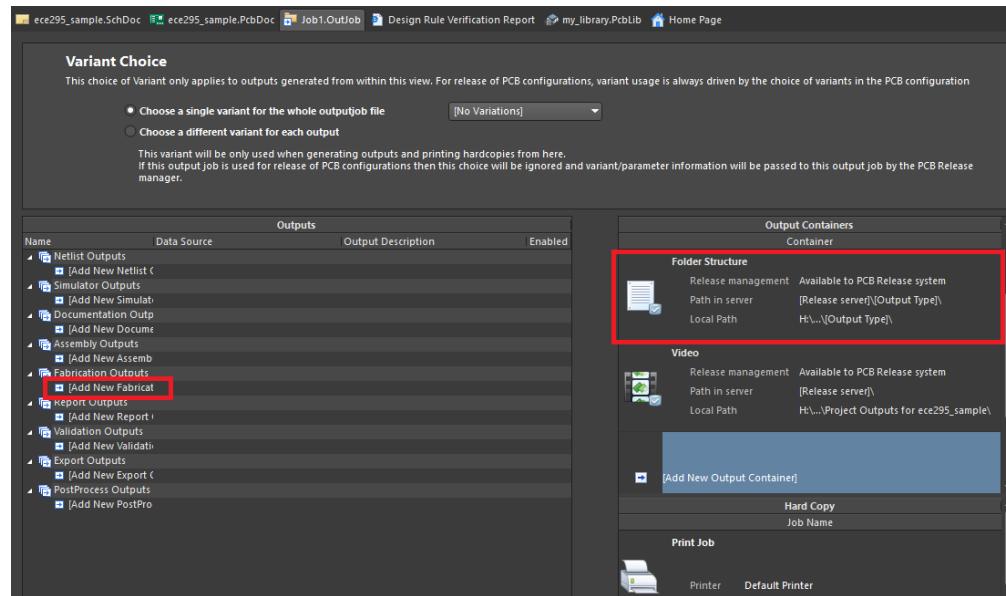
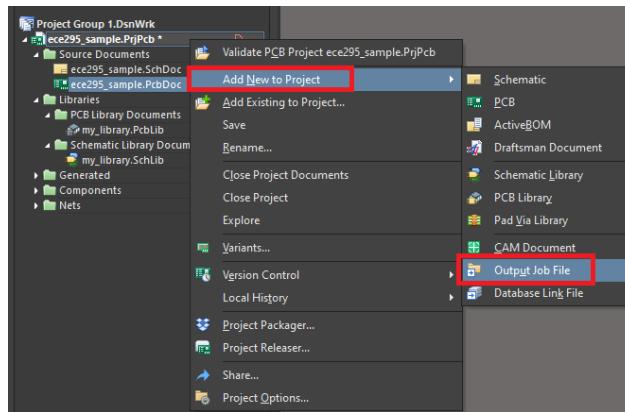
5 Preparing Files to Send For Manufacturing

Once a PCB has no errors or rule violations, the design is finished and the files can be prepared for manufacturing. As there are many possible tools for designing PCBs (Altium, Eagle, PowerPCB, ORCAD, Allegro, Kicad) it is not possible to simply send your Altium .PcbDoc file to a manufacturing company. Today, the industry standard file type for describing 2D PCB layers is called a **Gerber** file.

All PCB design tools support a way to output the design on a series of Gerber files and most allow these files to be viewed as well. There are also many free tools to view Gerber files.

5.1 Output Job

In Altium, the most streamlined way of preparing Gerber files is by adding an **Output Job File**. It can be added to a project by right-clicking and selecting **Add New To Project → Output Job File**.

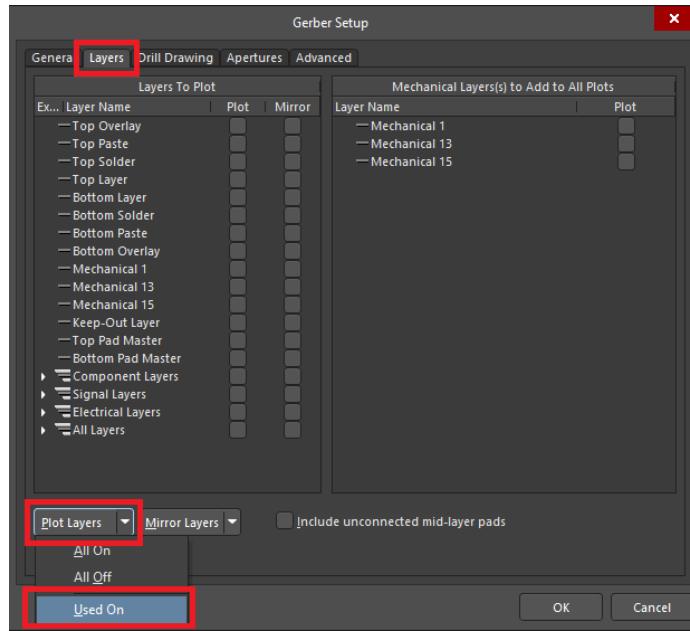


5.2 Gerber Files

Once the file is created, the outputs to generate must be added. Select **Add New Fabrication Output → Gerber Files → <your_PCB_name>.PcbDoc**. Double-click where it says **Gerber Files** to open the **Gerber Setup** form.

On the **Gerber Files** form navigate to the **Layers** tab at the top, then move to the bottom and select **Plot Layers**

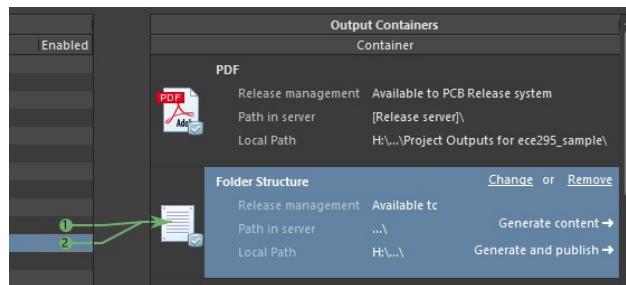
→ **Used On.** The board outline must be drawn on one of the selected layers, so ensure that is the case.



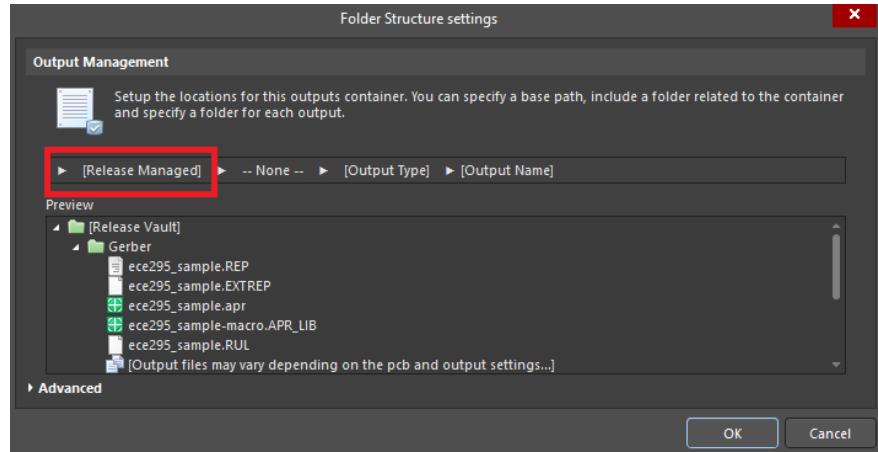
5.3 NC Drill Files

In addition to Gerber files, PCB manufacturers require **NC Drill Files**. These files specify the locations and sizes of all holes that need to be drilled into the PCB. To add them to the output, click **Add New Fabrication Output** → **NC Drill Files** → <your_PCB_name>.PcbDoc.

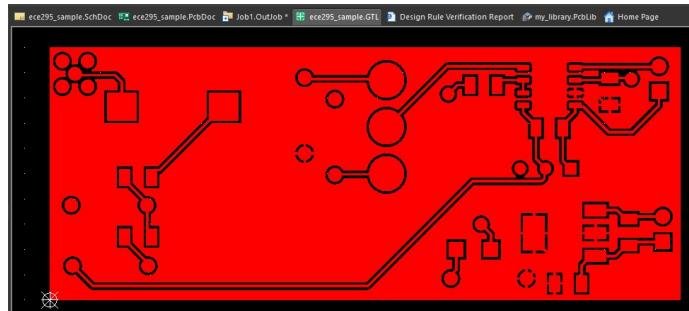
Enable both the Gerber and NC Drill file generation by clicking in the small ball under the **Enabled** column. It should turn green and point towards the **Output Containers** column.



The location of the outputs can be set under the **Output Containers** column. Under **Folder Structure** select **Change** to open the **Folder Structure settings** window. Click where it says **[Release Managed]**, change to **Manually Managed** and click **Done**. Then click **OK** to close the window.



To generate the requested content, simply press **Generate Content** ⇒ under the **Folder Structure** output container. After this, it is possible to open the generated Gerber files to view them in Altium. Shown below is the Gerber file for the sample design's top layer. After any changes to the design, the output files can be re-generated by pressing **Generate Content** ⇒ again.



6 Advanced Schematic Capture

6.1 Symbol Generation

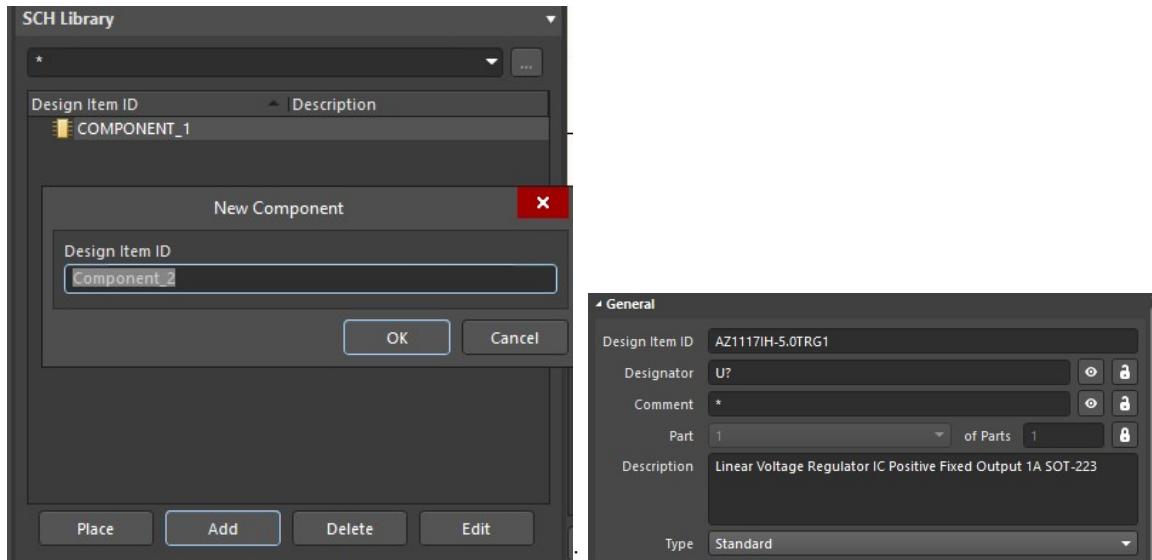
Sometimes you will find a part that you want to use, and Altium will not have a model available to place for the component. In this case, you can search for the part on a number of open-source websites, or you can make it yourself to include in one of your Altium libraries. Websites with part schematics and footprints include:

- SnapEDA <https://www.snapeda.com/>
- Ultra Librarian <https://www.ultralibrarian.com/>
- Component Search Engine <https://componentsearchengine.com/>
- Circuit Maker <https://circuitmaker.com/Components>

See Appendix B for more details about using these services.

To make your own part, you will need a library to include it in. To create a library, see Section 2.6. Once you have a library open, go to the **SCH Library** panel that opens on the bottom left and select “Add”. This brings up a window where you enter the ID of the new symbol, typically this should be the part number. We will use the same part from the Appendix B example, **AZ1117IH-5.0TRG1**

Right click or go to **Panels** to open the **Properties** menu and fill in the *Designator* and *Description* information.



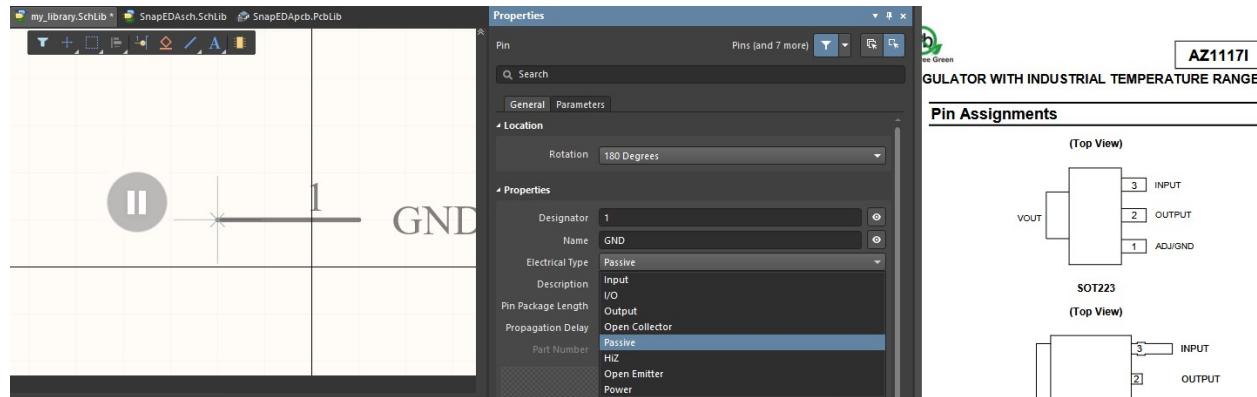
Typical designator letter names are as follows:

- | | |
|-------------------|--|
| • R? → Resistors | • Q? or M? → Transistors |
| • C? → Capacitors | • D? → LEDs and Diodes |
| • L? → Inductors | • P? or J? → Connectors |
| • S? → Switches | • U? → Generic Integrated Circuits (ICs) |

The question mark is a variable that will become a number depending on how many items of a given type are on your schematic.

You can draw basically anything you want for the schematic symbol, however the convention is for generic ICs to typically be a rectangle with inputs on the left and outputs on the right. The important thing is to create the correct number of **Pins** and ensure their numbers match the PCB footprint, and the datasheet.

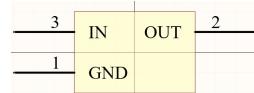
To make a pin, select **Place** → **Pin** and hit **TAB** to edit pin properties. The *Designator* number should match with the and the *Name* should be a short-form of whatever the pin is named in the datasheet.



If you select *Electrical Type*, you get a drop-down of pin types you can select to tell Altium more about the part and help Altium verify your design. Select the appropriate option, or leave it as passive if you are lazy/want to generate a bunch of warnings later.

Note 3. Pay attention to the direction of a pin. The connection point of a pin is determined by a small grey cross-hair attached to the mouse cursor and by four small white points on the placed pin. If you place the pin with the wrong connection point direction, it will be impossible to connect a wire to it.

After you have placed all of your pins, you should draw a rectangle around them with **Place** → **Rectangle**. If it covers the pins, go to *Properties* and check *Transparent*.



6.2 Hierarchy

In more complex designs, or designs with multiple copies of the same sub-circuit it is advantageous to create a hierarchy. This means there is one “top-level” schematic that contains multiple sub-circuits. In Altium, **Ports** are used to change between levels of the hierarchy.

Creating a hierarchy is not needed for the level of design complexity in this course. For completeness, the links below from Altium describe the process and why it is beneficial in many cases.

- Altium documentation on multi-sheet design
- Altium Webinar on Schematic hierarchy
- Altium Webinar: Tips and tricks for schematic hierarchy

6.3 Net Classes

Altium uses **Net Classes** to provide groupings between different nets. This can be useful to distinguish between nets that are part of a high-speed signal bus, power nets, or nets with high-voltage. Using **Net Classes** it is possible to apply different PCB layout rules for each class.

A complete description of how to setup and use Net Classes can be found in Altium's documentation [here](#).

A Guide to selecting parts from Digi-Key

First go to the Canadian Digi-Key website <https://www.digikey.ca/>

You should have a general idea of the function you need a part to perform, which should be the basis for your search. In this example we will look for an amplifier, so we type **amplifier** into the search bar.

If the search is broad, we will have different categories to choose from:

The screenshot shows the Digi-Key search interface with the search term 'amplifier' entered. Below the search bar, it says 'Top Results Showing 6 of 15'. There are six categories displayed in a grid:

- INTEGRATED CIRCUITS (ICs) Linear - Amplifiers - Instrumentation, OP Amps, Buffer Amps**: 34,313 items
- RF/F AND RFID RF Amplifiers**: 18,177 items
- INTEGRATED CIRCUITS (ICs) Linear - Amplifiers - Audio**: 5,182 items
- DEVELOPMENT BOARDS, KITS, PROGRAMMERS Evaluation Boards - Audio Amplifiers**: 748 items
- INTEGRATED CIRCUITS (ICs) Linear - Amplifiers - Special Purpose**: 1,633 items
- DEVELOPMENT BOARDS, KITS, PROGRAMMERS Evaluation Boards - Op Amps**: 1,066 items

A 'Show More' button is at the bottom right.

We want an audio amplifier in this case, so we select that box. All of the filtering options are available at the top, with the filtered parts in a table at the bottom.

The screenshot shows the Digi-Key search interface with the search term 'amplifier' entered. The top section has several filter panels highlighted with a red border:

- MANUFACTURER**: Includes a search filter and a list of manufacturers like AKM Semiconductor Inc., ams, Analog Devices Inc., Burr Brown, Cirrus Logic Inc., Dialog Semiconductor GmbH, Diodes Incorporated, and Elantec.
- SERIES**: Includes a search filter and a list of series like Acoustar™, NOVALOAD™, AudioMAX™, Automotive, AEC-Q100, and Automotive, AEC-Q100, Boomer®.
- PACKAGING**: Includes a search filter and a list of packaging types: -, Bag, Bulk, Cut Tape (CT), Dig-Reel®, Strip, Tape & Box (TB), Tape & Reel (TR), and Tray.
- PART STATUS**: Includes a search filter and a list of part statuses: Active, Discontinued at Digi-Key, Last Time Buy, Not For New Designs, Obsolete, and Preliminary.
- TYPE**: Includes a search filter and a list of part types: -, Class A, Class AB, Class B, Class H, Class D, Class AB, Class D, Class G, Class D, Class H, and Class D, Class AB.
- OUTPUT TYPE**: Includes a search filter and a list of output types: 1 Jack, Filtered, 1 Channel (Mono) or 2 Channel (Stereo), 2 Channel (Mono) with Mono Headphones, 1 Channel (Mono) with Mono and Stereo Headphones, 1 Channel (Mono) with Stereo Headphones, 2 Channel (Mono), 2 Channel (Stereo) or 4 Channel (Quad), and 2 Channel (Stereo) with Mono and Stereo Head.

The main search area shows '5,182 Results' and a 'SEARCH ENTRY' field with 'amplifier'. At the bottom, it shows 'Showing 1 - 25 of 5,182' and a table of results with columns: Compare, Mfr Part #, Price, Stock, Supplier, Mfr, Min Qty, DK Part #, Series, and Package. The first three rows are highlighted with a red border:

Compare	Mfr Part #	Price	Stock	Supplier	Mfr	Min Qty	DK Part #	Series	Package
<input type="checkbox"/>	TLC2472ID AUDIO AMPLIFIER	\$18.8550	500 - Immediate	Rochester Electronics, LLC	Rochester Electronics, LLC	16	2166-TLC2472ID-ND	-	Bulk
<input type="checkbox"/>	LM48556TL AUDIO AMPLIFIER/S	\$2.2329	500 - Immediate	Rochester Electronics, LLC	Texas Instruments	136	2166-LM48556TL-ND	Boomer®	Bulk
<input type="checkbox"/>	LM4850LD AUDIO AMPLIFIER	\$0.6154	33,655 - Immediate	Rochester Electronics, LLC	National Semiconductor	491	2166-LM4850LD-ND	Boomer®	Bulk

Note 4. Some important filter parameters must be selected each search to ensure your parts are possible to order

1. Select **In Stock** components

STOCKING OPTIONS

- In Stock (1,847)
- Normally Stocking (1,870)
- New Product (10)

2. Limit the **PACKAGING** options

PACKAGING

- Bag
- Bulk
- Cut Tape (CT)
- Digi-Reel®
- Strip
- Tape & Box (TB)
- Tape & Reel (TR)
- Tray
- Tube

The options that should not be selected are designed for use in large volume operations where machines select components from boxes or reels to be automatically placed on the PCB prior to soldering.

3. Set a **Pricing Quantity** (likely in the range of 1-10)

Showing 1 - 25 of 5,182 | Pricing Quantity

Compare	Mfr Part #	Price	Stock	Supplier	Mfr	Min Qty
---------	------------	-------	-------	----------	-----	---------

After selecting the three required options, you can start to filter your search further. At this point, many of the filtering headings will be specific to the type of part you need. For the audio amplifier, this includes selecting options like the type (Class A, Class AB, Class D, etc.), the output (1-Channel, 2-Channel, Stereo, Mono), as well as voltage and power requirements.

While you may not know all of these details, you might know some, or even some things that you might want to exclude.

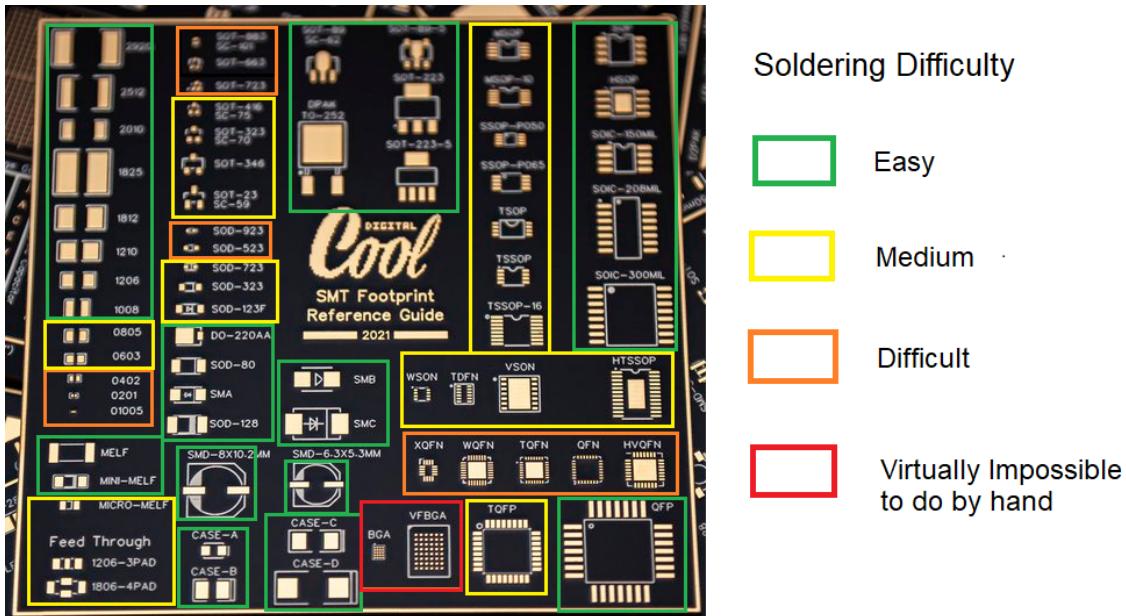
In this example, we want a Class AB or Class B amplifier with 1-Channel output that can operate off of a 5V supply. These details should come from your research and design steps.

TYPE	OUTPUT TYPE	MAX OUTPUT POWER X CHANNELS @ LOAD	VOLTAGE - SUPPLY
<input checked="" type="checkbox"/> Class AB <input checked="" type="checkbox"/> Class B <input type="checkbox"/> Class D <input type="checkbox"/> Class D, Class AB <input type="checkbox"/> Class D, Class G <input type="checkbox"/> Class D, Class H <input type="checkbox"/> Class DG <input type="checkbox"/> Class G	<input checked="" type="checkbox"/> 1-Channel (Mono) or 2-Channel (Stereo) <input checked="" type="checkbox"/> 1-Channel (Mono) with Mono Headphones <input checked="" type="checkbox"/> 1-Channel (Mono) with Mono and Stereo Headphones <input checked="" type="checkbox"/> 1-Channel (Mono) with Stereo Headphones <input checked="" type="checkbox"/> 1-Channel (Mono) <input type="checkbox"/> 2-Channel (Stereo) or 4-Channel (Quad) <input type="checkbox"/> 2-Channel (Stereo) with Mono and Stereo Headphones <input type="checkbox"/> 2-Channel (Stereo) with Stereo Headphones and Sub...	<input type="checkbox"/> 4mW x 2 @ 16Ohm <input type="checkbox"/> 4.5mW x 2 @ 30Ohm <input type="checkbox"/> 16mW x 2 @ 16Ohm <input type="checkbox"/> 17mW x 1 @ 40Ohm; 8.5mW x 2 @ 8Ohm <input type="checkbox"/> 23mW x 2 @ 16Ohm <input type="checkbox"/> 24mW x 2 @ 16Ohm <input type="checkbox"/> 25mW x 1 @ 16Ohm <input type="checkbox"/> 25mW x 2 @ 16Ohm	<input checked="" type="checkbox"/> 0V ~ 38V <input checked="" type="checkbox"/> 0V ~ 65V <input checked="" type="checkbox"/> 0V ~ 71.5V <input checked="" type="checkbox"/> 0V ~ 78V <input checked="" type="checkbox"/> ±0.02V ~ 75V <input checked="" type="checkbox"/> 0.9V ~ 1.8V <input checked="" type="checkbox"/> 0.9V ~ 2.5V

In theory at this point all of the presented options meet our design specifications, so it is time to pick one of the many available options (if no parts meet all of your specifications, circle back to the design and determine what

can be changed).

To narrow down remaining options an important consideration for manufacturability is the part footprint. The following image shows a selection of common surface mount part footprints with a guide on how difficult they are to solder by hand. Typically all through hole parts are easy enough to solder by hand, however many more surface mount parts are typically available.



To limit yourself to parts that are possible to use, on Digi-Key scroll to the rightmost sorting options.

 Do not select anything with **BGA** in the name, you will not be able to solder it

The easiest options to solder have DIP (through hole), SOP, SOIC, or QFP (surface mount) in the name.

Once you have limited the footprints, you can further narrow the available options by narrowing down your ranges, sorting by price, or stock. Start reading the datasheets of the top few components to get a better idea if the part will fit your application or not.

Most datasheets will show a diagram of a *Typical Circuit* or *Typical Application* for the IC, so you should look at that and ensure it makes sense with your design. You do not need to know how the chip functions, it can be somewhat of a black box, as long as you can understand the pins and how to interface with them. An example from the selected part, **TPA321**.

APPLICATION SCHEMATICS

Figure 26 is a schematic diagram of a typical handheld audio application circuit, configured for a gain of -10 V/V .

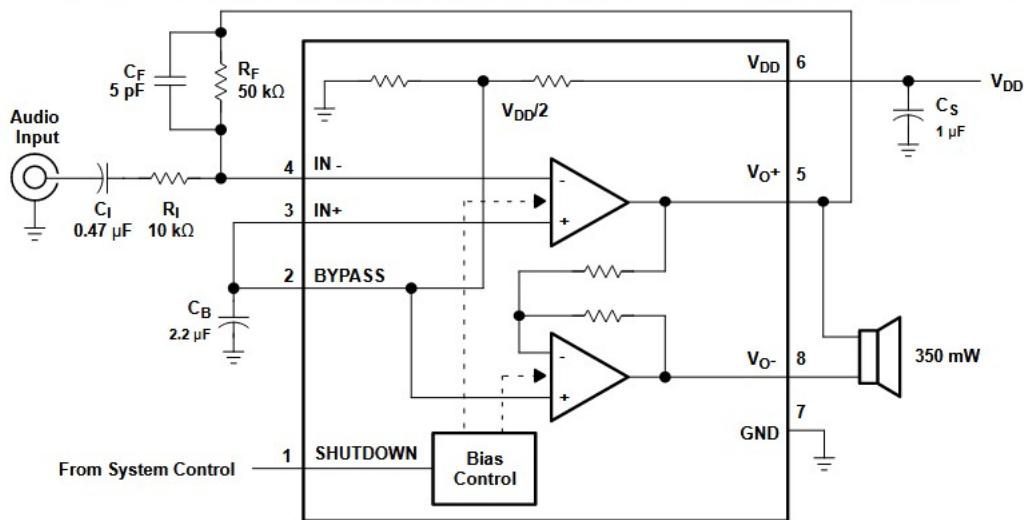


Figure 26. TPA321 Application Circuit

B Adding parts to your Library from 3rd Party Websites (SnapEDA Example)

Schematics and footprints are available for many parts through third party websites. Some common options for this include:

- SnapEDA <https://www.snapeda.com/>
- Ultra Librarian <https://www.ultralibrarian.com/>
- Component Search Engine <https://componentsearchengine.com/>
- Circuit Maker <https://circuitmaker.com/Components>

An account is typically needed to use these sites and it is best to follow along with their documentation, however a short guide on using SnapEDA is also included here.

Start with a part that you would like to use, but need either a schematic or footprint (or both) for. In this example, we've found that the power regulator LDO we want to use, AZ1117IH-5.0TRG1 does not have a symbol in Altium's Manufacturer Part Search.

We enter the part number, **AZ1117IH-5.0TRG1** into SnapEDA, and it returns one result. The symbols at the right indicate what is available for this part, in this case, the footprint is available, but the symbol for the schematic is not. At this point, there are two choices:

1. Make your own symbol (see Section 6.1)
2. Look harder

We can realize that the **-5.0TRG1** section of the part number is referring to the fact that this part has a fixed output of 5V, however there are other models with the same symbol and footprint that output different voltages.

Manufacturer	Image	Part	Package	Availability	Avg. Price (USD)	Description	Data Available
DIODES		AZ1117IH-5.0TRG1	SOT-223	<input checked="" type="checkbox"/>	\$0.14	Linear Voltage Regulator IC Positive Fixed 1 Output 1A SOT-223	<p>Datasheet available Symbol not available Footprint available 3D not available Sim not available</p>

Repeating our search using only **AZ1117IH** we see that the 3.3V version (-3.3TRG1) has the symbol and footprint available, and select that one.

Be careful doing something like this, be sure to double check the datasheets of the component and ensure the pin assignments match.

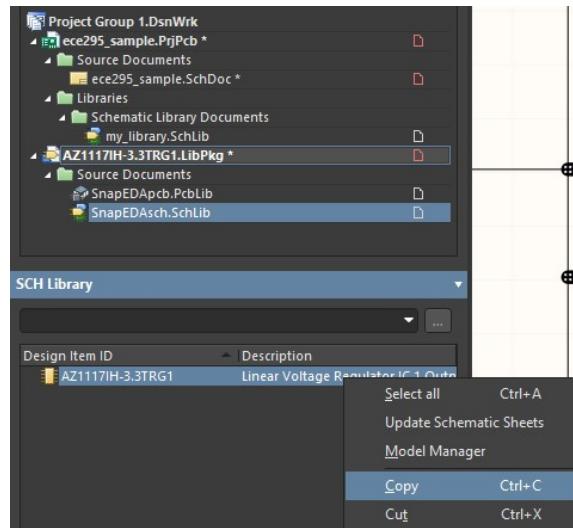
Click **AZ1117IH-3.3TRG1** then select **Download Symbol and Footprint → Altium**

You can then either *Extract Sources* or *Install Library*

- *Extract Sources* will allow you to open the library into the Projects window as an Integrated Library project.
- *Install Library* will install the library and add it to your Libraries Panel.

It is easiest to copy the designs into your own libraries through *Extract Sources*, but either option will give you access to the files you want.

To copy things into your library, double-click to open the newly extracted/imported library and then go to the item in the **SCH Library** panel on the left. Right click on the item and click copy (or CTRL+C). Then go to your library's **SCH Library** and right-click paste (or CTRL+V).



The same process applies to components in the **PCB Library**. See Section 3.2 for a description of adding parts to libraries.