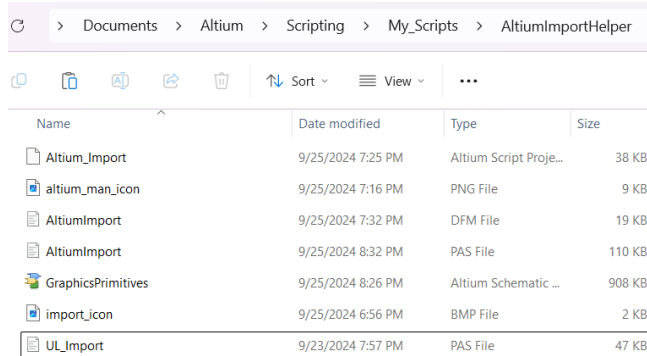


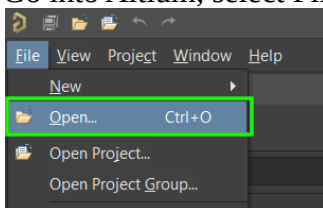
How To Install and Use Altium Import Helper

Installation

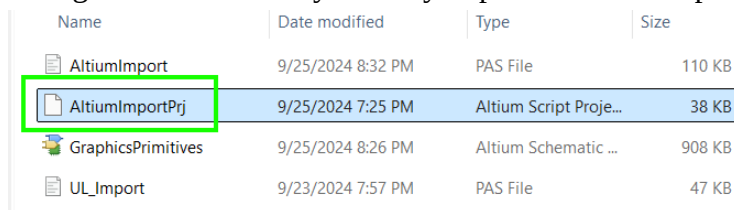
Place all the files into a permanent folder:



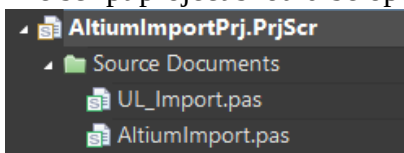
Go into Altium, select File → **Open**



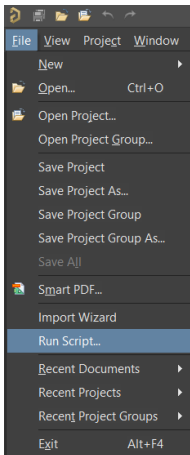
Navigate to the directory where you placed AltiumImportHelper, and open **AltiumImportPrj**



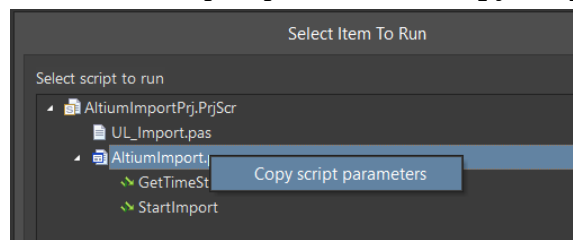
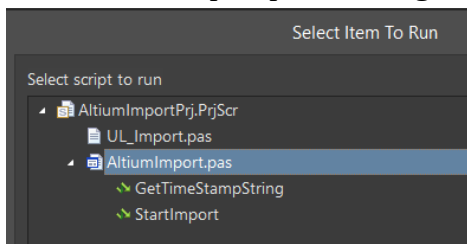
The script project should be open in Altium now



To install this script as part of the Altium GUI, Altium needs to know the script file and process to execute. To get this info, Click File → Run Script



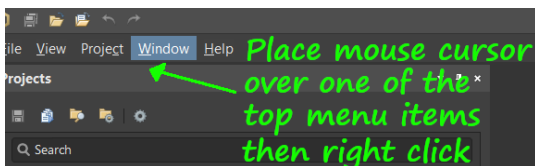
Select AltiumImport.pas, then **right click** over AltiumImport.pas and click **Copy script parameters**



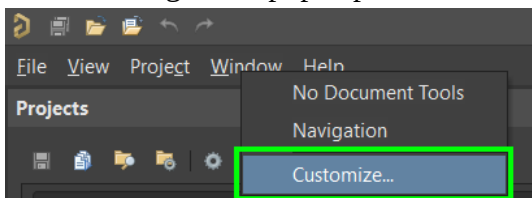
Then go into a word doc or something and paste it, so you can save it for later. It should produce something like this:

```
ProjectName=C:\Users\johnm\OneDrive\Documents\Altium\Scripting\My_Scripts\
AltiumImportHelper\AltiumImportPrj.PrjScr|ProcName=AltiumImport.pas>
```

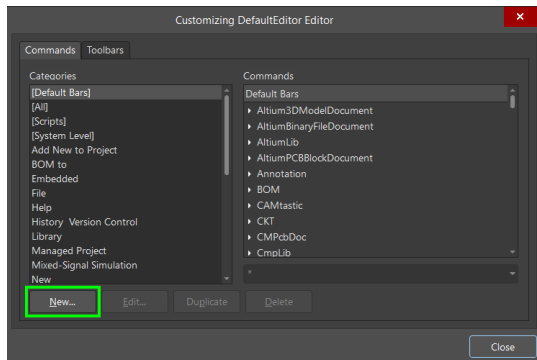
Exit the Altium Run Script dialog. Now we can add the script as a command to use in Altium. Put your mouse cursor over one of the top Altium menu items, then right click.



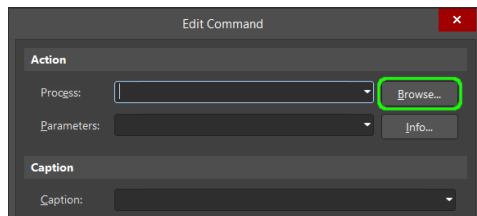
The following menu pops up, select **Customize...**



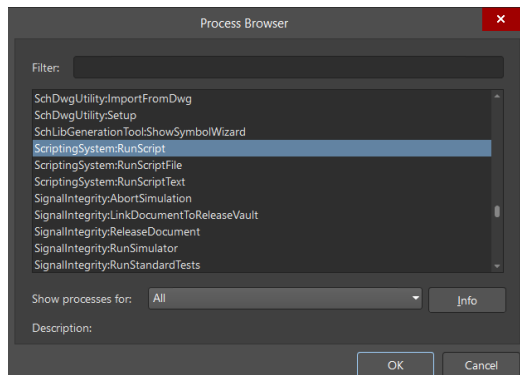
click 'New'



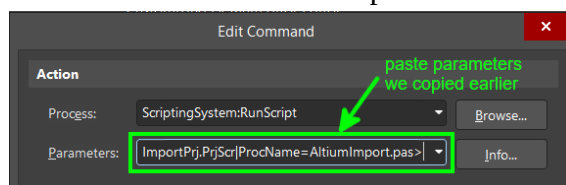
Click **Browse...**



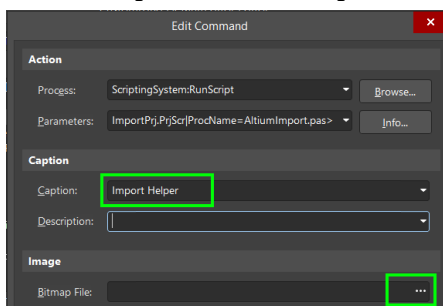
Scroll down and select **ScriptingSystem:RunScript**



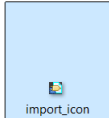
In the 'Parameters' text box paste the contents you saved in a word doc:



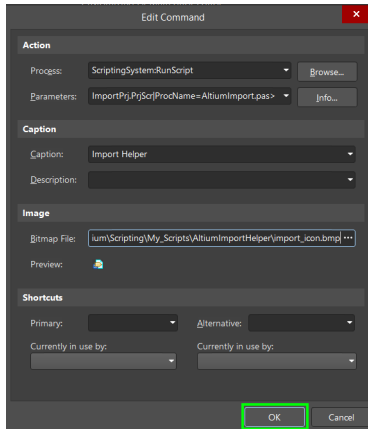
In the 'Caption' text box, put what you would like to appear when this command appears in a menu.



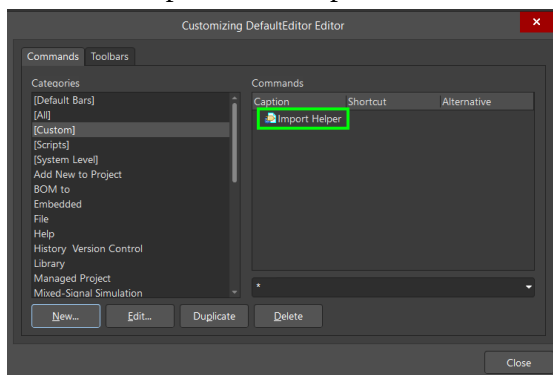
Click the Bitmap File button and navigate to where the script is stored, and select **import_icon**



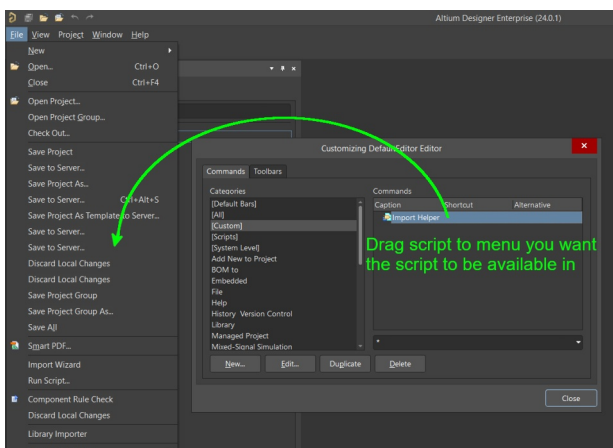
Then click Ok



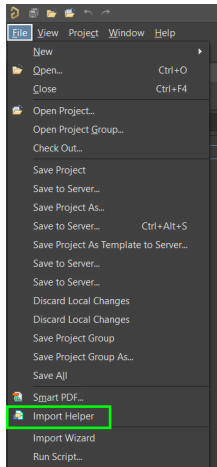
Now the script should be present in the 'Commands' box



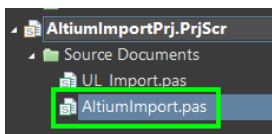
Next step to make the script easily accessible is to add the command to the GUI menus. The menus available in Altium are context sensitive, that is the menus available depend on what type of document is open. If you place the script in the File menu, it will always be available. To add the script to a menu, **drag it into the menu**



Then script will then show up in the menu.



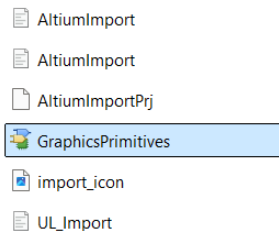
The last step is to configure the Import Helper for your use. With the script project still open in Altium, double click **AltiumImport.pas** to open it for editing.



Scroll down to around line 60, where you see the line **GRAPHICS_LIB_PATH = ''**;

```
*****  
- { CONSTANTS, EDIT ACCORDING TO YOUR PREFERENCE }  
-  
- const GRAPHICS_LIB_PATH = '';  
- const APPLY_FONTS_AND_COLORS_TO_SYMBOLS = True; // set this to True if you wa  
60 const PIN_FONT_NAME = 'Arial';  
- const PIN_FONT_SIZE = 9;  
- const DEFAULT_FONT = 'Arial';  
- const DEFAULT_FONT_SIZE = 9;  
- const COLOR_TEXT = $B00000; // BGR format  
- const COLOR_BLUE = $FF0000; // BGR format  
- const COLOR_BLACK = $000000; // BGR format  
- const COLOR_SYMBOL_OUTLINE = COLOR_BLUE;  
- const COLOR_SYMBOL_BODY_FILL = $B0FFFF; // light yellow, BGR format  
- const SHOW_MANUFACTURER_PART_NUMBER = True;
```

Now go back to the folder where you installed Altium Import Helper, select **GraphicsPrimitives** and **right click**. Select **copy as path**



Go back to Altium and paste the path into **GRAPHICS_LIB_PATH**, in between the quote ticks. After pasting, delete the double quotes. The red part that is **“C:\Users... and the part that is .SchLib”**

```
TANTS, EDIT ACCORDING TO YOUR PREFERENCE }  
GRAPHICS_LIB_PATH = "C:\Users\johnm\OneDrive\Documents\Altium\Scripting\My_Scripts\AltiumImportHelper\GraphicsPrimitives.SchLib";  
APPLY_FONTS_AND_COLORS_TO_SYMBOLS = True; // set this to True if you want below fonts and colors to take effect  
PIN_FONT_NAME = 'Arial';
```

After deleting the double quotes it should look like this:

```
GRAPHICS_LIB_PATH = 'C:\Users\johnm\OneDrive\Documents\Altium\Scripting\My_Scripts\AltiumImportHelper\GraphicsPrimitives.SchLib';
```

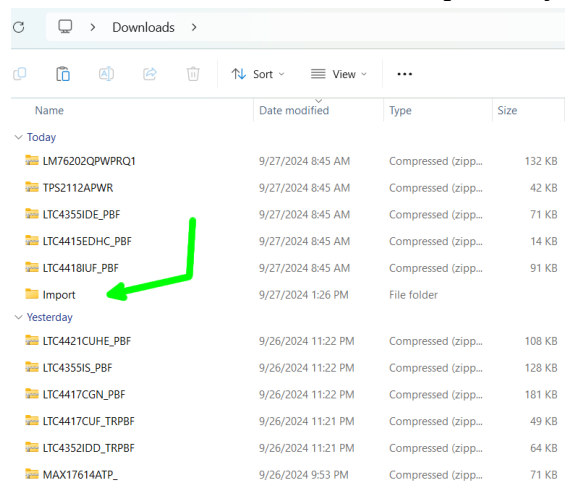
Save AltiumImport.pas.

How to Use Altium Import Helper

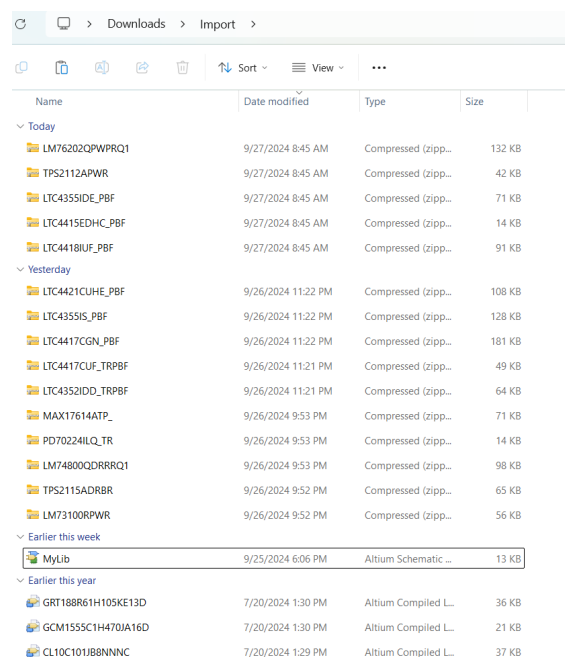
The script will automatically import almost any type of Altium file:

- UltraLibrarian zip file
- SnapEda zip file
- IntLib
- SchLib
- PcbLib

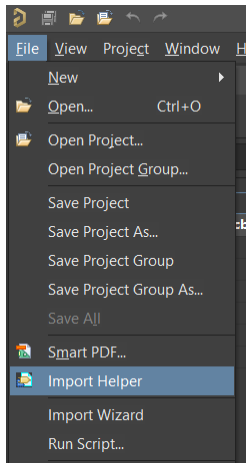
It is not necessary to unzip the files, the script will do that. All files will be imported into one LibPkg. First create a folder to place all files in. This is the same place where the LibPkg will be created. For instance create a folder called 'Import' in your Downloads dir.



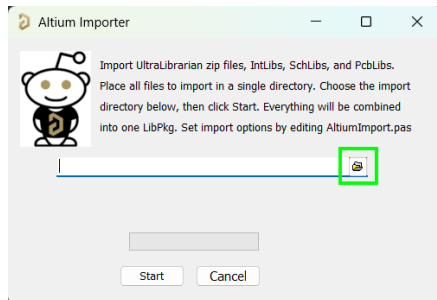
Move all the files into the 'Import' directory. Put any SchLibs, Intlibs, or PcbLib in the folder too.



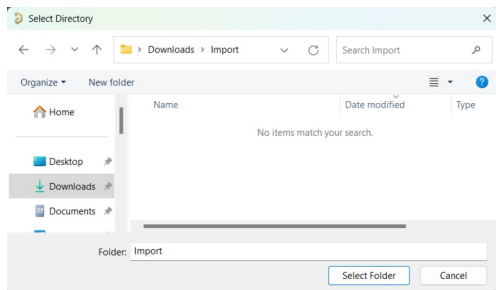
Go back to Altium, and run the script from the menu location where you placed it



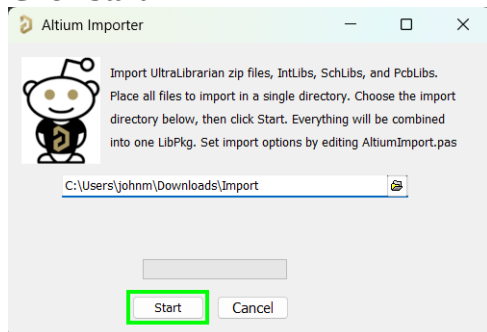
Click the folder icon to select the import location



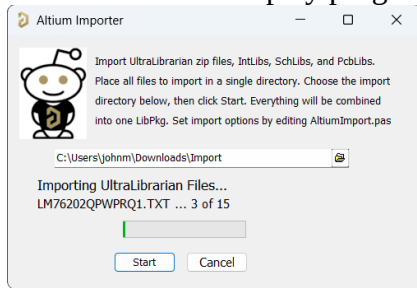
Select the Import folder where you placed the zip files to import. Note: you will not see the files in this window, only directories.



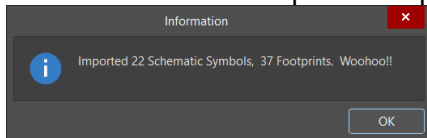
Click Start



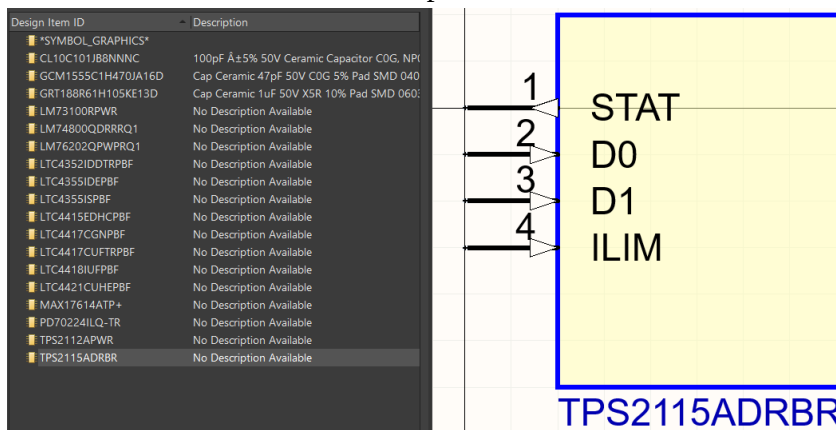
The window will display progress



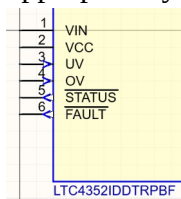
When finished the script will display how many symbols and footprints were imported



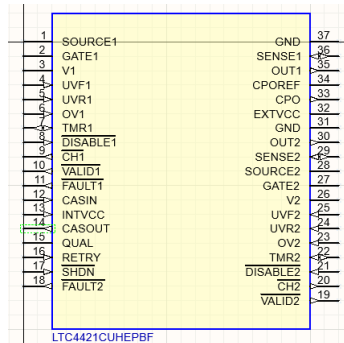
The SchLib and PcbLib will be open.



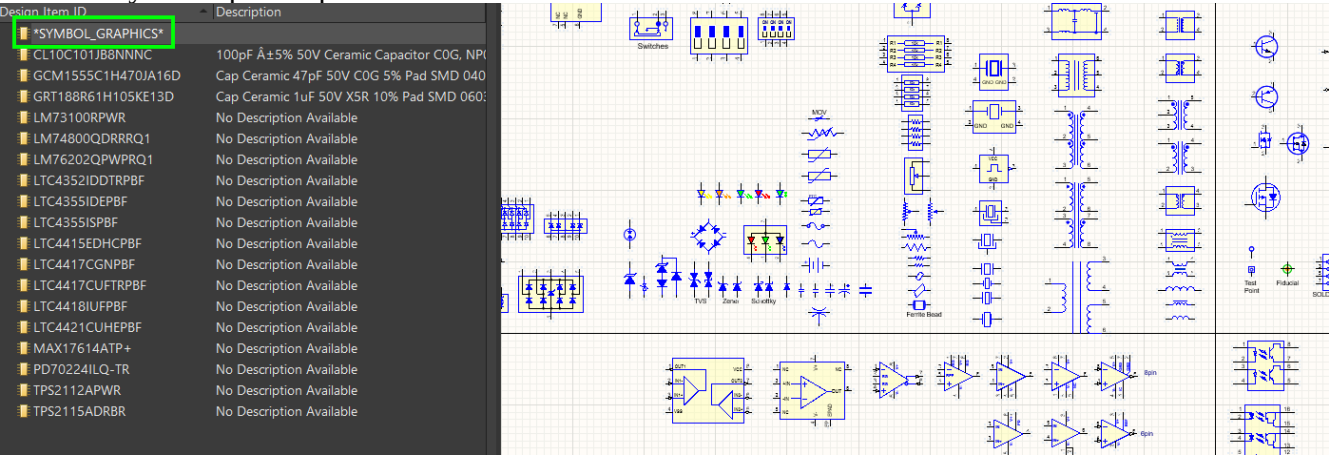
All schematic symbols will be normalized with the same fonts and colors. Active low pins will be appropriately displayed with an overbar. All symbols will have pin 1 at the origin.



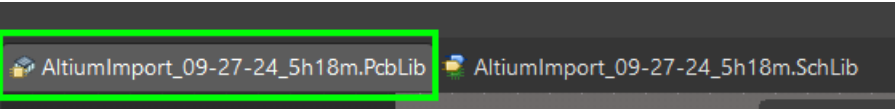
Manufacturer Part # will be placed at the bottom left hand corner of the symbol



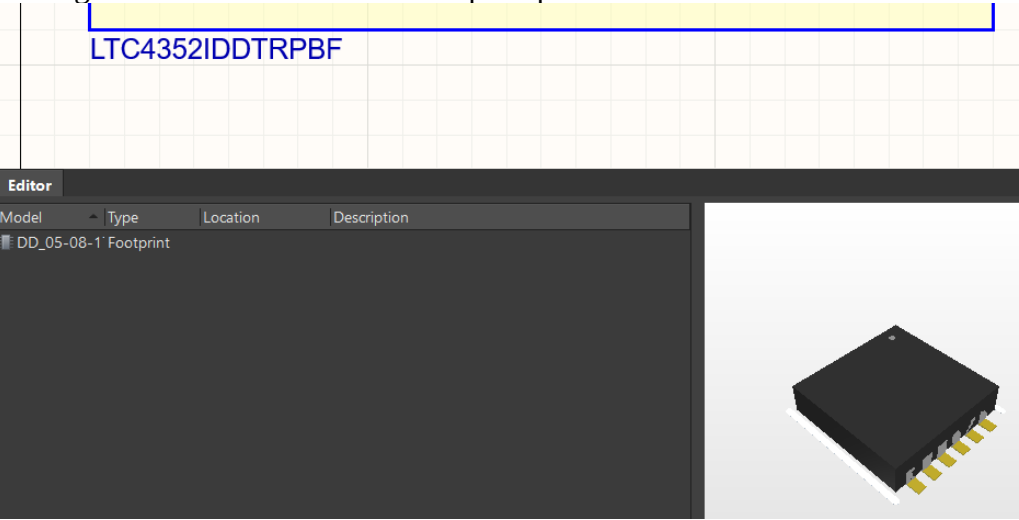
At the top of the SchLib is a symbol, ***SYMBOL_GRAPHICS***, use this to copy any graphics you want into your imported parts.



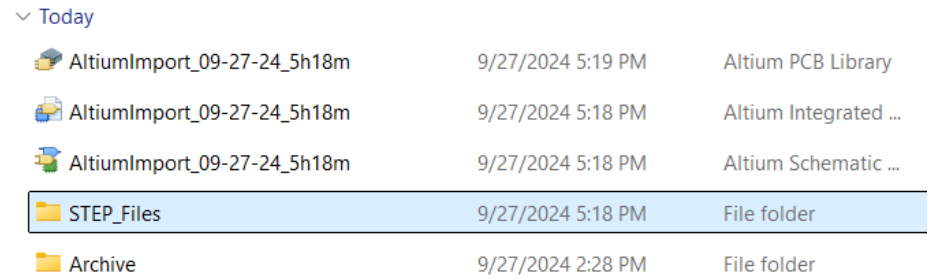
To get the footprint previews to be visible in the SchLib, click on the PcbLib so footprint view is visible, and save the PcbLib.



Then go back to the SchLib and footprint previews will be visible



If you go back to the import dir, where you placed the zip files to import, you'll find the created libraries, and also STEP_FILES dir and Archive dir.



All the STEP files are saved in case you need them later. All the zip files and IntLibs are placed in the Archive dir, so you can retrieve and re-import later if necessary.

Customizing The Script

At the top of AltiumImport.pas, you modify any of these constants to customize schematic symbols to your liking.

```
const APPLY_FONTS_AND_COLORS_TO_SYMBOLS = True; // set this to True if
const PIN_FONT_NAME = 'Arial';
const PIN_FONT_SIZE = 9;
const DEFAULT_FONT = 'Arial';
const DEFAULT_FONT_SIZE = 9;
const COLOR_TEXT = $B00000; // BGR format
const COLOR_BLUE = $FF0000; // BGR format
const COLOR_BLACK = $000000; // BGR format
const COLOR_SYMBOL_OUTLINE = COLOR_BLUE;
const COLOR_SYMBOL_BODY_FILL = $B0FFFF; // light yellow, BGR format
const SHOW_MANUFACTURER_PART_NUMBER = True;
```