Altium Documentation setup

# Preparing the Altium Project:

These are the requirements of an Altium PCB project in order for the documentation script to work as intended, some of these tasks may seem trivial for simple boards but it important that they all be done. The project contained in the test folder satisfies all of these requirements so can be used as a reference.

# Templates:

Please ensure that the ‘Pumpkin BOM Template with pricing.xlt’ and ‘Pumpkin Template.SchDot’ files from the templates folder are copied into the Altium templates folder at ‘C:\Users\Public\Documents\Altium\ADxx\Templates’

# Project Level:

1. The Project should have 45 project parameters:  
    AssemblyNumber: The PCBA number of the board. Eg. 710-01920  
    PartNumber: The PCB number of the board. Eg. 705-01573  
    ProjectRevision: eg. B0  
    AssemblyRevision: eg. B0  
    ProjectTitle: The name of the project to appear on the schematics.
2. The project should have two named variants:  
    Placed Components Only: All components to be placed for ASSY\_REV=0  
    SMD Assembly: Only the components to be machine placed  
   All DNP components in these Variants should be set as ‘Not Fitted’
3. The project should refer to the ‘Pumpkin Outjob.OutJob’ file in the ‘src’ directory. This document ensures that the outputs generated by the project satisfy the requirements of the documentation script.
4. All parts in the project need a valid link to a library component to ensure that the BOMs populate correctly.
5. The Project must contain a BOMDoc file to be used for BOM generation.

# Schematic documents:

1. All schematic documents must be designed on the ‘Pumpkin template.SchDot’
2. Each Schematic sheet must have two specific parameters populated to populate the title block:  
    DrawnBy: The initials of the designer eg DJW  
    Title: The Title of the sheet.
3. The project must include a modifications sheet (use the ‘MOD.SchDoc’ file in the templates folder as an example). This has a couple of requirements:
   1. The filename must be unchanged
   2. In the ASSY revision codes table every entry must end with a semicolon ‘;’.
   3. There should be no other semicolons present on the page, they are key characters that the code looks for.
4. Use the Page numbering tool to set the page numbers and the total sheet count.

# PCB document:

Use one of the ‘Pumpkin xxx Template.PcbDoc’ files in the templates folder as a reference.

1. All layers must be labelled somewhere on the layer with a text string with the layer name as per the reference document. If you use the template, this is already done for the layers that are already present.
   1. The ‘Bottom Silkscreen’ label must be mirrored.
   2. The ‘Bottom Soldermask’ label must be both mirrored and not mirrored.
   3. Fields such as .PartNumber will be filled when outputs are generated
   4. Layers should be numbered as per their order from the top side of the board to the bottom side.
   5. If you have more than 10 layers, or are using mechanical layers in an unusual fashion it may cause problems in the code. Contact david@asteriec.com if that is going to be an issue.
2. Refer to the Altium Standards document here for correct layer usage: <https://docs.google.com/document/d/1vNnC9ifGPp2p4NB-CMYa7VNW0V9QAhQonp9LnxyBmqo>
3. A drill table should be included.
4. Fabrication notes should be copied off the reference document and then adjusted to meet your requirements. These must be on the Mechanical 2 (Fab Notes) layer.
5. Add overall board dimensions to the Mechanical 2 (Fab Notes) layer.
6. The board outline should be on the Mechanical 1 (Board) layer.
7. Ensure a design rule is present for Board Outline Clearance.

Library Parts:

1. All library parts should have a valid supplier link (two is much preferred) where possible. If this is not possible then the component parameters need to be set to mirror that:
   1. Manufacturer 1: The Manufacturer of the component
   2. Manufacturer Part Number 1: The Manufacturer’s part number
   3. Supplier 1: The supplier to purchase from
   4. Supplier Part Number 1: The supplier’s part number
2. If parts have no assembly associated with them, eg. Test points, then the component type should be set to ‘Standard (No BOM)’

# Outputs:

In order to generate outputs from the project open the .Outjob file associated with the project

1. For each output container click the ‘generate content’ button.
2. For the Hard copy output, click the print button and save the file as ‘layers’ with the default xps extension. This file should be saved into the project directory.
3. If any outputs are highlighted in red, then something in your project needs to be addressed.

# Preparing the Documentation script:

1. If you haven’t already, clone the repository from  
   <https://github.com/PumpkinSpace/Altium_docs>
2. Ensure that you have Python 2.7 installed, if not go here:  
   <https://www.python.org/ftp/python/2.7.14/python-2.7.14.msi>
3. From the repository’s root folder run Altium\_Setup.py. This will check all of the project’s dependencies and install them as required. Some steps may involve intervention from the user to get things working. *This is especially true given this is a Python 2.7 application no Python 3 so there are potentially issues with the pip module*
4. When step 3 has ended with “setup successful!” copy the new Documentation.bat into the Altium project directory that you want to package.
5. Running this batch file will package the Altium project it is contained within.

# Obtaining google sheet credentials:

1. Follow this tutorial: <http://gspread.readthedocs.io/en/latest/oauth2.html>
2. Then copy the .json file that you downloaded into ‘src’ directory of the Altium Documentation repository