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 should 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 a copied into the Altium templates folder at 'C:\Users\Public\Documents\Altium\AD16\Templates'

Project Level:

1. The Project should have 4 project parameters:
   AssemblyNumber: The PCBA number of the board. Eg. 710-01920
   PartNumber: The PCB number of the board. Eg. 705-01573
   ProjectRevision: eg. B
   ProjectTitle: The name of the project to appear on the schematics.
2. The project should have a configuration called 'Placed Components Only' where all components that could be DNP are set to 'Not Fitted'. This includes all Through-hole components. ie. This is the list of components to be placed by a machine.
3. The project should include a copy of the 'Pumpkin Outjob.OutJob' file from the templates 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.

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 'BBI - MOD.SchDoc' file in the templates folder as an example). This has a couple of requirements:
   a. In the ASSY revision codes table every entry must end with a semicolon ';'.
   b. There should be no other semicolons present on the page, they are eky characters that the code looks for.
   c. Do not include the title 'ASSY' on any other sheet, it is a keyword that the code looks for.
4. Use the Page numbering tool to set the page numbers and the total sheet count.
5. Do not use the word 'of' in notes on the schematic sheet or in the title block. This is a keyword that the code looks for to assign page numbers.

PCB document:

Use the 'Pumpkin Template.PcbDoc' file 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.
   a. The 'Bottom Silkscreen' label must be mirrored.
   b. The 'Bottom Soldermask' label must be both mirrored and not mirrored.
   c. All labels bust be True Type in the Consolas font.
   d. Fields such as .PartNumber will be filled when outputs are generated
   e. Layers should be numbered as per their order from the top side of the board to the bottom side.
   f. If you have more than 10 layers it may cause problems in the code. Contact david@asteriec.com if that is going to be an issue.
2. A drill table should be included.
3. Fabrication notes should be copied off the reference document and then adjusted to meet your requirements. These must be on the mechanical 2 layer, do not change the font.
4. Add overall board dimensions to the Mechanical 2 layer.

Library Parts:

1. All library parts should have a valid supplier link where possible. If this is not possible then the component parameters need to be set to mirror that:
   a. Manufacturer 1:   The Manufacturer of the component
   b. Manufacturer Part Number 1:   The Manufacturer's part number
   c. Supplier 1:   The supplier to purchase from (optional)
   d. Supplier Part Number 1:   The supplier's part number (optional)
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.

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.
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 reposatory