PCB Validation logic

For Fab we need:

1. Gerber
2. Drill layers (this can be on Gerber, Excellon ASCII or Excellon EIA). Prefer is Exellon ASCII
3. Fab Drawings (this can be on Gerber, DXF, DWG, PDF).
4. Layer Sequence (this can be, readme, Fab Drawing, Gerber Header)
5. ODB++ (This could contain all the above)

For assembly we need:

1. All of the above
2. BOM
3. Assembly Drawings (this can be ODB++, gerber, PDF, DXF, DWG)
4. Paste Layers (this can be gerber or ODB++)
5. X&Y

Fab Logics:

1. Gerber. Parse the file a look for lines starting with %FSLA, %MOIN, %ADD, %LPD, G01, G75, G54, D02, D03, G36, G04
2. Drill Layers:
    a. Gerber. You can not tell by the content of the file. You can only look at the file name.
    b. Excellon ASCII. Parse the file a look for lines starting with M48, M72, T#C, T##C, G90, INCH,, FMAT,, G90
    Where “#” is any number (0-9) and “,,” is one comma.
    c. Excellon EIA. Is the same as ASCII. But EIA is a binary format. So when parsing the file the Binary need to be translated to HEX and HEX to the ASCII Character. Here is how it look on my Perl Code:

```
%AsciiCharacters = ("10" => " ", "01" => "1", "02" => "2", "04" => "4", "07" => "7",
"08" => "8", "10" => ":", "13" => "3", "15" => "5", "16" => "6", "19" => "9", "20" => "0",
"23" => "T", "25" => "V", "26" => "W", "29" => "Z", "31" => "\/", "32" => "S", "34" =>
"U", "37" => "X", "38" => "Y", "43" => "L", "45" => "N", "46" => "O", "49" => "R", "51"
=> "J", "52" => "K", "54" => "M", "57" => "P", "58" => "Q", "61" => "A", "62" => "B",
"64" => "D", "67" => "G", "68" => "H", "70" => "\+", "73" => "C", "75" => "E", "76" =>
"F", "79" => "I", "80" => "\n", "0b" => "\%", "0e" => "\&", "1a" => "\(", "3b" => ",", "3e"
=> "\t", "4a" => "\)", "5b" => "\%", "6b" => "\.", "6d" => "\@");
```

```
open DRL, "DrillLayer";
while (<DRL>) {@HexEIA = unpack '(H2)*', $_}
close DRL;

# Change EIA code to Character and check
##########################################################################
###############

$NewLine = "";
$XC = 1;
foreach $Code (@HexEIA) {

        chomp;

        if ($Code eq "80") {

                if ($NewLine =~ /M48/ or $NewLine =~ /M72/ or $NewLine =~ /
^T..C/ or $NewLine =~ /^T.C/ or $NewLine =~ /^G90/ or $NewLine =~ /^INCH,/ or
$NewLine =~ /^FMAT,/ or $NewLine =~ /^G90/) {$IsDrill = "yes"}
                if ($XC >= 20) {last}
                $XC++;
                $NewLine = "";

        } else {$NewLine = "$NewLine$AsciiCharacters{$Code}"}

}
```

3. Fab Drawings:
    a. Gerber you will have to look at the file name.
    b. DXF or DWG, I have never tried to Parse these type of files since they are not an ASCII format, so unless you can figure out how to do this and if they contain ASCII characters for the text you can look for FAB inside them or just look at the name.
    c. For PDF you look at the file name and do OCR.
4. Layer sequence. If it is gerber created by Altium you can parse the file and look for “G04 Layer_Physical_Order=2*”. Otherwise, good luck.
5. ODB++. ODB++ is a directory structure with ASCII files in them (for the most part). They are normally compress into one file.
    a. The correct way of compressing ODB++ is by creating a TAR and GZIP the TAR. But customers may create just a regular ZIP.
    b. After uncompressing the data, you will have the root directory that will have the name of the PCB or whatever the customer decides to call it.
    c. Inside the root folder as a minimum you will have an “steps” and “matrix” directory. Once you see these 2 directories you can tell this is an ODB++.
    d. Inside the matrix directory there will be a matrix file. This is the file that you can look at to see what layers the customer has supplied.

e. The matrix file is like xml. Bracket delimetered. You can parse this to see what the customer has provided. You will need to look at the LAYER Sections (Below is an example).

```
LAYER {
   ROW=1
   CONTEXT=BOARD
   TYPE=COMPONENT
   NAME=COMP_+_TOP
   POLARITY=POSITIVE
   START_NAME=
   END_NAME=
   OLD_NAME=
}

LAYER {
   ROW=2
   CONTEXT=BOARD
   TYPE=SILK_SCREEN
   NAME=TOPOVERLAY
   POLARITY=POSITIVE
   START_NAME=
   END_NAME=
   OLD_NAME=
}

LAYER {
   ROW=3
   CONTEXT=BOARD
   TYPE=SOLDER_PASTE
   NAME=TOPPASTE
   POLARITY=POSITIVE
   START_NAME=
   END_NAME=
   OLD_NAME=
}

LAYER {
   ROW=4
   CONTEXT=BOARD
   TYPE=SOLDER_MASK
   NAME=TOPSOLDER
   POLARITY=POSITIVE
```

START_NAME=
END_NAME=
OLD_NAME=
}

LAYER {
ROW=5
CONTEXT=BOARD
TYPE=SIGNAL
NAME=TOP
POLARITY=POSITIVE
START_NAME=
END_NAME=
OLD_NAME=
}

LAYER {
ROW=6
CONTEXT=BOARD
TYPE=SIGNAL
NAME=BOTTOM
POLARITY=POSITIVE
START_NAME=
END_NAME=
OLD_NAME=
}

f. Layer types signal, mixed or power_ground and the context is board. These will be the copper layers. By looking at the row number you can calculate the layer sequence and layer count.

g. Other layer types with context board will define mask, silk, paste and drill layers.

h. Layer types with context misc will most likely be the drawing and the name may define what type of drawing.

i. Layers types called component maybe used as assembly drawings.

Assembly Logic:

1. BOM. For the most part they are excel format. In some cases they are PDF. You can Identify this by file name and parsing or OCR and look for the following key words: ITEM|SN|MANUF|MPN|PARTNUM|MANFPART|MFG|MANFP|MFRP|MFGR|MP#|VENDOR|VPN|DISTRIBUTOR|DPN|DIGIKEY|DISTRIBUTOR_P|QUANTITY|REFERENCE|DESIGNATOR|

REFDES|SCHEMATIC|DONOTINSTALL|DNI|DNP|DONOTPOPULATE|STUFF|DESC|LIBREF|PARTDESCRIPTION

2. Assembly drawings. This is going to be the same as Fab drawing, but different keywords like assem.

3. Paste Layers. This can be found thru the FAB logic.

4. X&Y. I'm not very familiar with this, I believe they can be ASCII files or excel. I do not believe that there is a standard format for this. Somebody in assembly will have to give more input.