The Gerber Job Format Specification

Revision 2020.08

Including fabrication documentation in Gerber

This format was developed by Karel Tavernier , Ucamco.

Contents

Copyright Ucamco NV   2

Having a question or remark about the spec? Please contact us at gerber@ucamco.com

1 Preface

Since decades, Gerber is the standard for describing the 2D images – copper layers, solder mask, drills – in PCB fabrication data. Attributes added with Gerber X2 provide a standard to 'add intelligence to the image' by defining the layer structure – which file is which layer – identify via and SMDs component, pin numbers and reference descriptions.

But PCB fabrication data is not just about images - it includes finish, overall thickness, materials, solder mask color. They are essential for the quoting, planning, engineering, CAM and fabrication of the bare and assembled board. It is often transferred informally, in drawings and texts, and handled manually, wasting time and risking errors.

This specification defines how to transfer this information in a machine-readable manner as part of Gerber fabrication data.

Please send questions or suggestions about this specification to gerber@ucamco.com

Copyright Ucamco NV

3

Having a question or remark about the spec? Please contact us at gerber@ucamco.com

2 Specification

2.1 Overview

PCB characteristics pertaining to the job as a whole, such as the finish or overall thickness, are stored in a separate file, the Gerber job file. One job file per PCB. It contains the following sections:

1. **Header**: information about the file itself, such as its creation date.

2. **General specs**: overall board characteristics such as finish.

3. **Material stackup:** specification of the stackup and characteristics of the materials used.

4. **Design rules**: the design rules used when laying out the PCB.

5. **Files attributes:** the polarity and function (e.g. top copper layer) of all Gerber files.

The Gerber job file is a separate file, not part of the image files. It contains only PCB characteristics and no image data. Applications that process a Gerber job file do not need to be image-savvy.

The image format is not changed. Thus, compatibility is achieved. Legacy applications can happily ignore the job file and process the images as before. Existing image input/output code does not need to be changed when implementing the Gerber job file.

The Gerber job file is compatible with both X1 and X2.

Partial implementations are allowed. Better a partial job file than no job file at all.

The job file can be extended with custom information.

The job file is simple and human readable.

The job file intentionally does *not* specify default values for the PCB characteristics. If a characteristic is not present on the job file, it is not defined *in* the job file and must be specified by other means, for instance fabricator defaults or a side agreement. This restriction is necessary to allows partial implementations: otherwise it would be unclear whether a missing parameter specifies the default of is simply not implemented. And anyhow, a mere data transfer format must not pretend to specify what defaults the PCB industry should use.

The job file intentionally defines the *technical* PCB parameters only, *not* commercial conditions such as delivery times, pricing, and quantities. The reason is simple. The same PCB can be ordered at different times, with different commercial conditions, or it can be ordered in a panel. The physical PCB however is the same, and its definition must not be touched. Commercial conditions are best in a separate file.

The job file intentionally does *not* cover assembly panels or arrays. Layout systems are designed to generate a single PCB. The panel is typically not decided by the designer but by the assembler. The same PCB may be assembled in different panels in different production runs, or multiple PCB's can be mixed on a single panel. Therefore, the assembly panel definition must be separate from the single PCB definition. However, the job file is important to the assembler as it contains essential information such as the board size.

2.2 Syntax

The job file follows the JSON syntax. For an introduction to JSON see http://json.org/ . For an online JSON syntax checker see http://jsoneditoronline.org/ .

The length unit is mm.

The Gerber Job File uses UTF-8 encoding.

Numbers must fit in the range of an IEEE double. For readability, number that are naturally integer, such as the layer count, must be written as integer, without decimal points.

The standard file extension is ".gbrjob". An example name: `Ctlr54382r4.gbrjob`

Partial implementations are allowed. One can omit entire sections, or e.g. define only a few board characteristics in the General Specs section. Missing characteristics are undefined, there are no defaults.

2.3 Header

This object has values that provide information about the Gerber job file itself rather than about the PCB it describes, e.g. the creation date of the file.

<table>
<tr><th colspan="2">Header</th></tr>
<tr><th>Name/value</th><th>Usage</th></tr>
<tr>
<td>

```
"GenerationSoftware": {
  "Vendor": "<string>",
  "Application": "<string>",
  "Version": "<string>
}
```

</td>
<td>GenerationSoftware identifies the software that generated the job file. "Vendor" identifies the organization that supplies the generating software to its users; this can be a commercial company, an open source organizatio, etc.

It is semantically identical to the .GenerationSoftware attribute in the Gerber file specification.</td>
</tr>
<tr>
<td>

`"CreationDate": "<string>"`

</td>
<td>Identifies the moment of creation of the Gerber file. The value must conform to the full version of the ISO 8601 date and time format, including the time zone.

It is semantically identical to the .CreationDate attribute in the Gerber file specification.</td>
</tr>
<tr>
<td>

`"Comment": "<string>"`

</td>
<td>Comment about the file.</td>
</tr>
</table>

Example:

```
"Header": {
  "GenerationSoftware": {
    "Vendor": "Ucamco",
    "Application": "UcamX",
    "Version": "2017.12"
  },
  "CreationDate": "2018-01-20T15:59:51+01:00",
  "Comment": "Example"
}
```

2.4 General Specs

This object contains characteristics that apply to the whole board, such as finish. Characteristics which apply to a specific element, such as the legend color, are stored in other objects where the subject such as legend is identified. The object and all characteristics below are optional.

<table>
<tr>
<th colspan="2">General Specs

(Unit is mm)</th>
</tr>
<tr>
<th>Name/value</th>
<th>Usage</th>
</tr>
<tr>
<td>

```
"ProjectId": {
  "Name":     "<string>",
  "GUID":     "<string>",
  "Revision": "<string>"
}
```

</td>
<td>

Usually a Gerber file is part of a PCB project with a sequence of revisions. The purpose of the .ProjectId file attribute is to uniquely identify project and revision. This is especially important to check whether all files belong to the same revision. By its nature, these values can only be defined by the creator of the project and revision.

"Name" is the id or reference used by the design owner, "GUID" defines the project using a global unique ID and "Revision specifies its revision. All parameters must conform to the string syntax, with the additional restriction that the ',' character cannot be used. The "GUID" value is a field conforming to RFC4122 version 1 or version 4.

It is semantically identical to the .ProjectId attribute in the Gerber file specification.

</td>
</tr>
<tr>
<td>

```
"Owner": "<string>"
```

</td>
<td>Reference of the design owner, as used by himself.</td>
</tr>
<tr>
<td>

```
"Size": {
  "X":    <number>,
  "Y":    <number>,
  "Tol+": <number>,
  "Tol-": <number>
}
```

</td>
<td>

Board size, being the size of the axis-aligned enclosing rectangle of the board outline, in mm. *It is strongly recommended to include this information in any job file.*

The optional tolerances in mm.

</td>
</tr>
<tr>
<td>

```
"LayerNumber": <integer>
```

</td>
<td>Number of copper layers. *It is strongly recommended to include this information in any job file.*</td>
</tr>
<tr>
<td>

```
"BoardThickness": <number>
```

</td>
<td>The overall thickness of the base material and all conductive materials deposited thereon, in mm.</td>
</tr>
<tr>
<td>

```
"IPC-2221-Type": <integer>
```

</td>
<td>

The board type according to IPC-2221. The integer can take the values from 1 to 6, corresponding to the six primary board types:
• Type 1 - Single-sided
• Type 2 - Double-sided
• Type 3 – Multilayer, TH components only
• Type 4 – Multilayer, with TH, blind and/or buried vias.
• Type 5 - Multilayer metal-core board, TH components only
• Type 6 - Multilayer metal-core

</td>
</tr>
</table>

`"IPC-600-Class": "`**`(1`**`|`**`2`**`|`**`3`**`|`**`NA)`**`"`	
`"Standard": <`**`string`**`>`	MIL, JSS, IPC6012B, PCA600, …
`"ImpedanceControlled": (`**`true`**`|`**`false`**`)`	
`"UL_Logo": (`**`true`**`|`**`false`**`)`	Indicates whether an UL logo must be added. The exact PCB requirements this corresponds to are left open at this time.
`"Fabricator_Logo": (`**`true`**`|`**`false`**`)`	Indicates whether a fabrication logo must be added.
`"Fabricator_Datecode": (`**`true`**`|`**`false`**`)`	Indicates whether a fabrication date code must be added.

`"ViaProtection": ["`**`<IPC-4761>`**`" {,"`**`<IPC-4761>`**`"}]`
`where`
`<IPC-4761>=(Ia|Ib|IIa|IIb|IIIa|IIIb|IVa|IVb|V|VI|VII|None)`
	This via protection types present, according to the IPC-4761 classification:
	Ia Tented - Single-sided
	Ib Tented - Double-sided
	IIa Tented and Covered – Single-sided
	IIb Tented and Covered – Double-sided
	IIIa Plugged – Single-sided
	IIIb Plugged – Double-sided
	IVa Plugged and Covered – Single-sided
	IVb Plugged and Covered – Double-sided
	V Filled (fully plugged)
	VI Filled and Covered
	VII Filled and Capped
	None No protection
	
	The protection type of each via tool is specified by .AperFunction in the Gerber drill files.

`"HolePlatingThickness": `**`<number>`**	The plating thickness in the holes.
`"HalogenFree": (`**`true`**`|`**`false`**`)`	
`"Press-fit": (`**`true`**`|`**`false`**`)`	Presence of press-fit holes. These are holes for connector pins that will not be soldered but pressed in the holes. They require tighter tolerances.
`"HeatSinkPaste": (`**`true`**`|`**`false`**`)`	Indicates the presence of heat sink paste.
`"EdgePlating": (`**`true`**`|`**`false`**`)`	
`"Castellated": (`**`true`**`|`**`false`**`)`	

Copyright Ucamco NV   8

Having a question or remark about the spec? Please contact us at gerber@ucamco.com

`"EdgeConnector": (`**`true`**`|`**`false`**`)`	
`"EdgeConnectorBevelled": (`**`true`**`|`**`false`**`)`	
`"HardGoldArea": `**`<number>`**	Area of hard gold in mm squared. Where that gold is can be defined with a gold mask file.
`"RoHS": (`**`true`**`|`**`false`**`)`	

`"Finish": "(`
`HASL SnPb|`
`HASL lead-free|`
`Immersion tin|`
`Immersion nickel|`
`Immersion silver|`
`ENIG|`
`ENEPIG|`
`Hard gold|`
`OSP|`
`HT_OSP|`
`None|`
`<string>`
`)"`	

`"Foil": "`**`(Electro-Deposited`** **`|Rolled|<string>))"`**	Copper foil type

`"Substrates": [ "`**`<substrate>`**`"`
`{,"`**`<substrate>`**`"}]`
`where `**`<substrate>`**` =`
`(FR4|Polyimide|Polyolefin|Al|PTFE|`
`Teflon|Ceramic|<string>)`
	The substrate(s) used in the PCB. The order of the substrates in this list is arbitrary and is not related to the physical order in the PCB.

`"Material_Tg": `**`<number>`**	Minimal glass transition temperature, in ºC.
`"ITAR": (`**`true`**`|`**`false`**`)`	The job is ITAR controlled.
`"ElectricalTest": (`**`true`**`|`**`false`**`)`	
`"Notes": "`**`<string>`**`"`	A free field with informal information.

Example:

```
"GeneralSpecs": {
  "Size": {
    "X": 160,
    "Y": 50.8
  },
  "LayerNumber": 4,
  "BoardThickness": 1.6,
  "IPC-600-Class": 2,
  "Finish": "ENIG"
}
```

Copyright Ucamco NV   10

Having a question or remark about the spec? Please contact us at gerber@ucamco.com

2.5 Material Stackup

The Material Stackup array defines the material layers and their attributes. For most standard designs the stackup is obvious and a full description is not needed. However even in simple design one may want to define the copper weights and solder mask and legend colors.

An array element is a layer in the Material Stackup. The order of the elements is the order of the material layers. The type of material is identified by the "Type:" pair and is mandatory for each array element. Other optional pairs specify properties of each material. Common sense indicates which properties are appropriate for which material.

If the Material Stackup is included, it must be complete - all layers of the PCB, must be present, and only those materials. Materials without any attributes can be added to indicate the presence of that material.

Material Stackup	
Name/value	**Name/value**
`"Type": "(Copper|Dielectric|Legend|SolderMask|CoverLay|PeelableMask|Carbon|`**`<string>`**`)"`	
`"Thickness":` **`<number>`**	*Base* thickness in mm. This does *not* include any extra thickness added by plating.
`"Color": "(Red|Yellow|Green|Blue|White|Black|R<integer>G<integer>B<integer>)[,(Gloss|Semi-matte|Matte)]"` `(`<integer> ranges from 0 to 255`)`	
`"DielectricConstant":` **`<number>`**	Dielectric constant.
`"LossTangent":` **`<number>`**	Loss tangent.
`"Conductivity":` **`<number>`**	Conductivity in S/m.
`"Tg":` **`<number>`**	Minimal glass transition temperature in ºC.
`"Material": "`**`<substrate>`**`"`	Material used as substrate.
`"Name": "`**`<string>`**`"`	A name for the layer. It has no board functionality, but it can be referred to in associated documents and be used to help in communication.
`"Note": "`**`<string>`**`"`	A note on the layer or material.

`"Substacks:" [`
 	
`<integer>{,<integer>}`
 	
`]`
	This array is only applicable for flex-rigid boards.
	
	It uses the concept of substacks that is used by Polar Instruments in its handling of flex-rigid boards. There is a master stackup, with all material present; it is substack 1. Different parts of the flex-rigid use different substacks, each identified by a positive integer. The array substacks contains the indices of the substacks in which the material is present, in ascending order.

An example shows best how it is done. Here is the material stackup of a four-layer board, with solder mask on both sizes and legend on top:

👁 **Example:**

```
"MaterialStackup": [
  {
    "Type":      "Legend",
    "Color":     "White"
  },
  {
    "Type":      "SolderMask",
    "Thickness": 0.025,
    "Color":     "Green"
  },
  {
    "Type":      "Copper",
    "Thickness": 0.03556
  },
  {
    "Type":      "Dielectric",
    "Thickness": 0.34
  },
  {
    "Type":      "Copper",
    "Thickness": 0.015
  },
  {
    "Type":      "Dielectric",
    "Thickness": 0.34,
    "DielectricConstant": 4.7
  },
  {
    "Type":      "Copper",
    "Thickness": 0.015
  },
  {
    "Type":      "Dielectric",
```

Copyright Ucamco NV

12

Having a question or remark about the spec? Please contact us at gerber@ucamco.com

```
      "Thickness": 0.34
    },
    {
      "Type":      "Copper",
      "Thickness": 0.03556
    },
    {
      "Type":      "SolderMask",
      "Thickness": 0.025,
      "Color":     "Green"
    }
  ]
```

Note that this describes a CAD stackup – also called virtual stackup by Polar Instruments. A layer represents a type of material, such as copper or dielectric. Two or three prepreg layers can be used to fabricate one dielectric layer, and a core can be used for two copper layers separated by a dielectric. In other words, in fabrication the virtual or CAD stackup is converted into a real fabrication stackup.

If we only need to specify the colors the material stackup is very simple, see the example below.

Example:

```
"MaterialStackup": [
  { "Type": "Legend",     "Color": "White" },
  { "Type": "SolderMask", "Color": "Green" },
  { "Type": "Copper"                       },
  { "Type": "Dielectric"                   },
  { "Type": "Copper"                       },
  { "Type": "Dielectric"                   },
  { "Type": "Copper"                       },
  { "Type": "Dielectric"                   },
  { "Type": "Copper"                       },
  { "Type": "SolderMask", "Color": "Green" }
]
```

One may remark that this requires plenty of dummy lines to represent just three colors. While true, this is not really a problem. The very generic stackup array allows to define the colors. It makes no sense to introduce a redundant ad-hoc structure dedicated to the colors, just to save a few dummy lines.

Copyright Ucamco NV   13

Having a question or remark about the spec? Please contact us at gerber@ucamco.com

2.6 Design Rules

The design rules array contains the intended design rules used by CAD to lay out the copper layers. Thus the intended design rules are known without needing to perform complex geometric analysis of the copper layers. This is useful information for planning and quoting. It helps CAM to detect and handle possible layout problems.

The copper layers are grouped in classes with the same design rules. Typically, design rules are different for inner and outer layers, and then the classes are inner and outer. Each design rule class is an entry in the array. The entry consists of a definition of the layers in the class and their design rules.

Design rules (Unit is mm.)	
`"Layers": "(Inner|Outer|L<integer>)"`	Design rule group. All layers in the group share the same design rules. The precedence is "Ln", "Inner" or "Outer". There is no inheritance, all applicable design rule values must be set for each class.
`"PadToPad": <number>`	
`"PadToTrack": <number>`	
`"PadToRegion": <number>`	Regions are also known as copper pours.
`"TrackToTrack": <number>`	
`"TrackToRegion": <number>`	
`"RegionToRegion": <number>`	
`"MinLineWidth": <number>`	
`"MinRing": <number>`	
`"MinClearanceToProfile": <number>`	
`"Notes": "<string>"`	Informal information.

An example with the typical inner/outer design rules shows how it works.

Example:

```
"DesignRules": [
  {
    "Layers": "Outer",
    "PadToPad":      0.15,
    "PadToTrack":    0.15,
    "TrackToTrack":  0.15,
    "MinLineWidth":  0.2,
    "TrackToRegion": 0.381,
    "RegionToRegion": 0.381
  },
  {
    "Layers": "Inner",
    "PadToPad":      0.15,
    "PadToTrack":    0.15,
    "TrackToTrack":  0.15,
    "MinLineWidth":  0.254,
    "TrackToRegion": 0.508,
    "RegionToRegion": 0.508
  }
]
```

Copyright Ucamco NV

15

Having a question or remark about the spec? Please contact us at gerber@ucamco.com

2.7 Files Attributes

This section contains meta-information about the files the fabrication data set, not actual PCB characteristics. This meta-information can be embedded in Gerber X2 files. However, in legacy formats it cannot. This section attaches meta-information to the legacy files without changing the files themselves. In an X2 context this section can still be useful as the meta-information is available centrally, without opening each Gerber image file.

The files attributes array attaches attributes to each Gerber file in the archive. Each array element contains the attributes associated with one file; the file is identified by its path from the location of the job file, which serves as a unique index for each array element. The order of the files in the array is free; for human readability it is suggested to sort them from top to bottom.

The following attributes are supported.

Name/value	Usage
`"Path": "<string>"`	The path to the file.
`"FileFunction": "<string>"`	This attribute has the same interpretation and can take the same values as the .FileFunction X2 attribute. See the Gerber File Format Specification. There is one exception where the attribute value can take a value that is not from the Gerber specification. In Gerber plated and non-plated drill holes must be put in separate files, so that plating can be identified at the file level. However, in legacy drill data plated and non-plated holes are sometimes lumped together in the same file. (Specifying plating is then complicated.) The file function value then becomes "XPlated,i,j,XTH", where i,j denotes the drill span. This value is not allowed for as a Gerber file attribute!
`"FilePolarity": "Positive|Negative"`	This attribute has the same interpretation and can take the same values as the .FilePolarity X2 attribute. See the Gerber File Format Specification. Take care, solder masks are normally negative.
`"FileFormat": "(Gerber|XNC|NC|SM|IPC356|Other)"`	Identifies the file format. Gerber Gerber, both X1 and X2 XNC XNC format (Exchange NC) NC IPC-NC-349 or Excellon SM Sieb & Meyer IPC356 IPC-D-356 Other Any other format

An example shows best how it is done.

Having a question or remark about the spec? Please contact us at gerber@ucamco.com

Ucamco

Example:

```
"FilesAttributes": [
  {
    "Path":         "AZ2375EM_Top_Silk.gbr",
    "FileFunction": "Legend,Top",
    "FilePolarity": "Positive",
    "FileFormat":   "Gerber"
  },
  {
    "Path":         "AZ2375EM_Top_SolderMask.gbr",
    "FileFunction": "Soldermask,Top",
    "FilePolarity": "Negative",
    "FileFormat":   "Gerber"
  },
  {
    "Path":         "AZ2375EM_Top_Copper.gbr",
    "FileFunction": "Copper,L1,Top",
    "FilePolarity": "Positive",
    "FileFormat":   "Gerber"
  },
  {
    "Path":         "AZ2375EM_Bot_Copper.gbr",
    "FileFunction": "Copper,L2,Bot",
    "FilePolarity": "Positive",
    "FileFormat":   "Gerber"
  },
  {
    "Path":         "AZ2375EM_Bot_SolderMask.gbr",
    "FileFunction": "Soldermask,Bot",
    "FilePolarity": "Negative",
    "FileFormat":   "Gerber"

  },
  {
    "Path":         "AZ2375EM_Drill.xnc",
    "FileFunction": "XPlated,1,4,XTH",
    "FileFormat":   "XNC"
  },
  {
    "Path":         "AZ2375EM_Test.ipc",
    "FileFunction": "Other",
    "FileFormat":   "IPC356"
  },
  {
    "Path":         "AZ2375EM_Drawing.gbr",
    "FileFunction": "FabricationDrawing",
    "FilePolarity": "Positive",
    "FileFormat":   "Gerber"
```

```
            }
        ]
```

2.8 JSON Schema

Below is a JSON schema. The schema prevails if there is a conflict with the narrative part of the specification. The schema is also published as a separate file on the website, for convenience. The schema was developed by Arezoo Kashefi.

```json
{
    "$schema": "http://json-schema.org/draft-07/schema#",
    "title": "JSON Schema for the Gerber Job File",
    "description": "Applies to revision 2020.08, see www.ucamco.com for
the full specification",
    "type": "object",
    "properties": {
        "Header": {
            "type": "object",
            "properties": {
                "GenerationSoftware": {
                    "type": "object",
                    "properties": {
                        "Vendor": {
                            "type": "string"
                        },
                        "Application": {
                            "type": "string"
                        },
                        "Version": {
                            "type": "string"
                        }
                    }
                },
                "CreationDate": {
                    "type": "string",
                    "anyOf": [
                        {
                            "format": "date-time"
                        },
                        {
                            "format": "date"
                        }
                    ]
                },
                "Comment": {
                    "type": "string"
                }
            },
            "additionalProperties": false
        },
        "GeneralSpecs": {
            "type": "object",
            "properties": {
```

```json
            "ProjectId": {
                "type": "object",
                "properties": {
                    "Name": {
                        "type": "string"
                    },
                    "GUID": {
                        "type": "string"
                    },
                    "Revision": {
                        "type": "string"
                    }
                }
            },
            "Owner": {
                "type": "string"
            },
            "Size": {
                "type": "object",
                "properties": {
                    "X": {
                        "type": "number"
                    },
                    "Y": {
                        "type": "number"
                    },
                    "Tol+": {
                        "type": "number"
                    },
                    "Tol-": {
                        "type": "number"
                    }
                },
                "required": [
                    "X",
                    "Y"
                ]
            },
            "LayerNumber": {
                "type": "integer"
            },
            "BoardThickness": {
                "type": "number"
            },
            "IPC-2221-Type": {
                "type": "integer",
                "enum": [
                    1,
```

```
                                        2,
                                        3,
                                        4,
                                        5,
                                        6
                                    ]
                            },
                            "IPC-600-Class": {
                                "type": [
                                    "string",
                                    "integer"
                                ],
                                "enum": [
                                    1,
                                    2,
                                    3,
                                    "NA"
                                ]
                            },
                            "Standard": {
                                "type": "string"
                            },
                            "ImpedanceControlled": {
                                "type": "boolean"
                            },
                            "UL_Logo": {
                                "type": "boolean"
                            },
                            "Fabricator_Logo": {
                                "type": "boolean"
                            },
                            "Fabricator_Datecode": {
                                "type": "boolean"
                            },
                            "ViaProtection": {
                                "type": "array",
                                "items": {
                                    "type": "string",
                                    "enum": [
                                        "Ia",
                                        "Ib",
                                        "IIa",
                                        "IIb",
                                        "IIIa",
                                        "IIIb",
                                        "IVa",
                                        "IVb",
                                        "V",
```

Copyright Ucamco NV

21

Having a question or remark about the spec? Please contact us at gerber@ucamco.com

```
                        "VI",
                        "VII",
                        "None"
                    ]
                }
            },
            "HolePlatingThickness": {
                "type": "number"
            },
            "HalogenFree": {
                "type": "boolean"
            },
            "Press-fit": {
                "type": "boolean"
            },
            "HeatSinkPaste": {
                "type": "boolean"
            },
            "EdgePlating": {
                "type": "boolean"
            },
            "Castellated": {
                "type": "boolean"
            },
            "EdgeConnector": {
                "type": "boolean"
            },
            "EdgeConnectorBevelled": {
                "type": "boolean"
            },
            "HardGoldArea": {
                "type": "number"
            },
            "RoHS": {
                "type": "boolean"
            },
            "Finish": {
                "type": "string"
            },
            "Foil": {
                "type": "string"
            },
            "Substrates": {
                "type": "array",
                "items": {
                    "type": "string"
                }
            },
```

Copyright Ucamco NV

22

Having a question or remark about the spec? Please contact us at gerber@ucamco.com

```
                "Material_Tg": {
                    "type": "number"
                },
                "ITAR": {
                    "type": "boolean"
                },
                "ElectricalTest": {
                    "type": "boolean"
                },
                "Notes": {
                    "type": "string"
                }
            },
            "additionalProperties": false
        },
        "MaterialStackup": {
            "type": "array",
            "items": {
                "type": "object",
                "properties": {
                    "Type": {
                        "type": "string"
                    },
                    "Thickness": {
                        "type": "number"
                    },
                    "Color": {
                        "type": "string",
                        "pattern":
"^(Red|Yellow|Black|Blue|Green|White|(R([01][0-9][0-9]|2[0-4][0-9]|25[0-
5])G([01][0-9][0-9]|2[0-4][0-9]|25[0-5])B([01][0-9][0-9]|2[0-4][0-9]|25[0-
5])))(?=((, ?Gloss|, ?Semi-matte|, ?Matte)?)$)"
                    },
                    "DielectricConstant": {
                        "type": "number"
                    },
                    "LossTangent": {
                        "type": "number"
                    },
                    "Conductivity": {
                        "type": "number"
                    },
                    "Tg": {
                        "type": "number"
                    },
                    "Material": {
                        "type": "string"
                    },
                    "Name": {
```

Copyright Ucamco NV   23

Having a question or remark about the spec? Please contact us at gerber@ucamco.com

```json
                "type": "string"
            },
            "Note": {
                "type": "string"
            },
            "Substacks": {
                "type": "array",
                "items": {
                    "type": "integer"
                }
            }
        },
        "minProperties": 1,
        "required": [
            "Type"
        ]
        },
        "additionalItems": false
    },
    "DesignRules": {
        "type": "array",
        "items": {
                        "type": "object",
        "properties": {
            "Layers": {
                "type": "string",
                "oneOf": [
                    {
                        "enum": [
                            "Inner",
                            "Outer"
                        ]
                    },
                    {
                        "pattern": "^L[0-9]+$"
                    }
                ]
            },
            "PadToPad": {
                "type": "number"
            },
            "PadToTrack": {
                "type": "number"
            },
            "PadToRegion": {
                "type": "number"
            },
            "TrackToTrack": {
```

Copyright Ucamco NV

24

Having a question or remark about the spec? Please contact us at gerber@ucamco.com

```
                        "type": "number"
                    },
                    "TrackToRegion": {
                        "type": "number"
                    },
                    "RegionToRegion": {
                        "type": "number"
                    },
                    "MinLineWidth": {
                        "type": "number"
                    },
                    "MinRing": {
                        "type": "number"
                    },
                    "MinClearanceToProfile": {
                        "type": "number"
                    },
                    "Notes": {
                        "type": "string"
                    }
                },
                "required": [
                    "Layers"
                ],
                "minProperties": 2,
                "additionalProperties": false
                },
                "additionalItems": false
        },
        "FilesAttributes": {
            "type": "array",
            "items": {
                "type": "object",
            "properties": {
                "FileFunction": {
                    "type": "string"
                },
                "FilePolarity": {
                    "type": "string",
                    "enum": [
                        "Positive",
                        "Negative"
                    ]
                },
                "Path": {
                    "type": "string"
                },
                "FileFormat": {
```

```
                    "type": "string"
                }
            },
            "required": [
                "Path"
            ]
            },
            "additionalItems": false
        }
    },
    "additionalProperties": false
}
```

3 **Examples**

3.1 Minimal CAD Job File

Below is the minimal job file that CAD must include in the fabrication data. CAD systems 'know' board size and layer count. CAD systems must output a simple job file with board size and layer count (and ideally any other parameters they might know). This is the basis for pricing and allowing assemblers to define their panels automatically without having to process Gerber image files. Other applications can read and extend this initial job file with more information.

```
{
  "Header": {
    "GenerationSoftware": {
      "Vendor":      "Ucamco",
      "Application": "UcamX",
      "Version":     "2017.12"
    },
    "CreationDate":  "2018-01-20T15:59:51+01:00",
    "Comment": "Example"
  },
  "GeneralSpecs": {
    "Size": {
      "X": 160.0,
      "Y":  50.8
    },
    "LayerNumber":    4,
    "BoardThickness": 1.6
  }
}
```

Having a question or remark about the spec? Please contact us at gerber@ucamco.com

3.2 Basic Job File

The job file contains the basic board parameters needed for a quote next to the images.

```
{
  "Header": {
    "GenerationSoftware": {
      "Vendor":       "Ucamco",
      "Application": "UcamX",
      "Version":     "2017.12"
    },
    "CreationDate":  "2018-01-20T15:59:51+01:00",
    "Comment":       "Example"
  },
  "GeneralSpecs": {
    "Owner": "Galactic Corporation",
    "ProjectId": {
      "Name":      "HFDF Controller nRF52832",
      "GUID":      "f81d4fae-7dec-11d0-a765-00a0c91e6bf6",
      "Revision": "Rev. 2b"
     },
    "Size": {
     "X": 160.0,
     "Y":  50.8
    },
    "LayerNumber":    4,
    "BoardThickness": 1.6,
    "ROHS":           true,
    "IPC-600-Class":  2,
    "Finish":         "ENIG"
  },
  "MaterialStackup": [
    { "Type": "Legend",      "Color":      "White" },
    { "Type": "SolderMask",  "Color":      "Green" },
    { "Type": "Copper",      "Thickness": 0.03556 },
    { "Type": "Dielectric"                        },
    { "Type": "Copper",      "Thickness": 0.01500 },
    { "Type": "Dielectric"                        },
    { "Type": "Copper",      "Thickness": 0.01500 },
    { "Type": "Dielectric"                        },
    { "Type": "Copper",      "Thickness": 0.03556 },
    { "Type": "SolderMask",  "Color":      "Green" }
  ]
}
```

4 **Revisions**

Rev 2020.08

The .Path name/value was put in the table of 2.7, thanks to Bruce McKibben for pointing out it was only in the example.

Several clarifications. Added JSON schema, see 2.8

Rev 2020.01

Removed the "Part:" name/value pair. Removed custom names. Restricted the file to UTF-8 encoding. Clarified "GenerationSoftware:", "CreationDate:" and "ProjectId:". Fixed text errors pointed out by Nick Meeker. Text improvements.

Rev 2019.09

Correction in values for finishes in 2.4: replaced 'HAL' by 'HASL' and 'Immersion gold (ENIG)' by 'ENIG'. Add the "Name:" to the material stackup name/value pairs, as requested by Jean-Pierre Charras, see section 2.5.

Rev 2019.05

Correction in example 3.2, pointed out by Arezoo Kashefipour.

Rev 2019.03

Added "FileFormat:" name/value pair to handle files in other formats than Gerber, see section 2.7; Jean-Pierre Charras contributed to this enhancement. Small text corrections.

Rev 2019.02

Deprecated superfluous design rule group All and General Specs value .Part.

"Foil" values contained <f>, replaced by <string>.

"Type" values contained Other <string>, replaced by <string>

Rev 2018.04

This is the initial version of the Gerber Job File specification.

This specification was developed by Karel Tavernier. An initial draft was published for public review by the Gerber user community in January 2017. Based on the review input the draft went through seven revisions. The review process was closed in April 2018 and resulted in specification 2018.04. Jean-Pierre Charras, Wim De Greve, Alan Shrives, Bruce McKibben, Ken Caluwaert, Aurelio Bantigue, Luc Samyn, Scott McClusky, Richard Attrill, Paul Wells-Edwards, Remco Poelstra, Rik Breemeersch contributed to this specification.

Copyright Ucamco NV   29

Having a question or remark about the spec? Please contact us at gerber@ucamco.com

5 Copyright

© Copyright Ucamco NV Gent Belgium

Ucamco owns copyrights in this document. All rights reserved. No part of this document or its content may be re-distributed, reproduced or published, modified or not, translated or not, in any form or in any way, electronically, mechanically, by print or any other means without prior written permission from Ucamco. One reason Ucamco must retain its copyrights in the Gerber Format® specification is to maintain the integrity of the standard.

The information contained herein is subject to change without prior notice. Revisions may be issued from time to time. This document supersedes all previous versions. Users of the Gerber Format®, especially software developers, must consult www.ucamco.com to determine whether new revisions were issued.

Ucamco developed the Gerber Format®. All intellectual property contained in it is solely owned by Ucamco. By publishing this document Ucamco has not granted any license or alienated any rights to the intellectual property contained in it. To use this intellectual property, developers of software interfaces based on this format specification must make a reasonable effort to comply with the latest revision of the specification; this is necessary to maintain the integrity of the standard.

Gerber Format® is a Ucamco registered trademark. By using this document, developing software interfaces based on this format or using the name Gerber Format®, users agree not to (i) rename the Gerber Format®; (ii) associate the Gerber Format® with data that does not conform to the Gerber file format specification; (iii) develop derivative versions, modifications or extensions without prior written approval by Ucamco; (iv) make alternative interpretations of the data; (v) communicate that the Gerber Format® is not owned by Ucamco, explicitly or implied.

The material, information and instructions are provided AS IS without warranty of any kind, explicit or implied. Ucamco does not warrant, guarantee or make any representations regarding the use of the information contained herein, or the results of its use. Ucamco shall not be liable for any direct, indirect, consequential or incidental damages arising out of the use or inability to use the information contained herein.

All product names cited are trademarks or registered trademarks of their respective owners.

Having a question or remark about the spec? Please contact us at gerber@ucamco.com