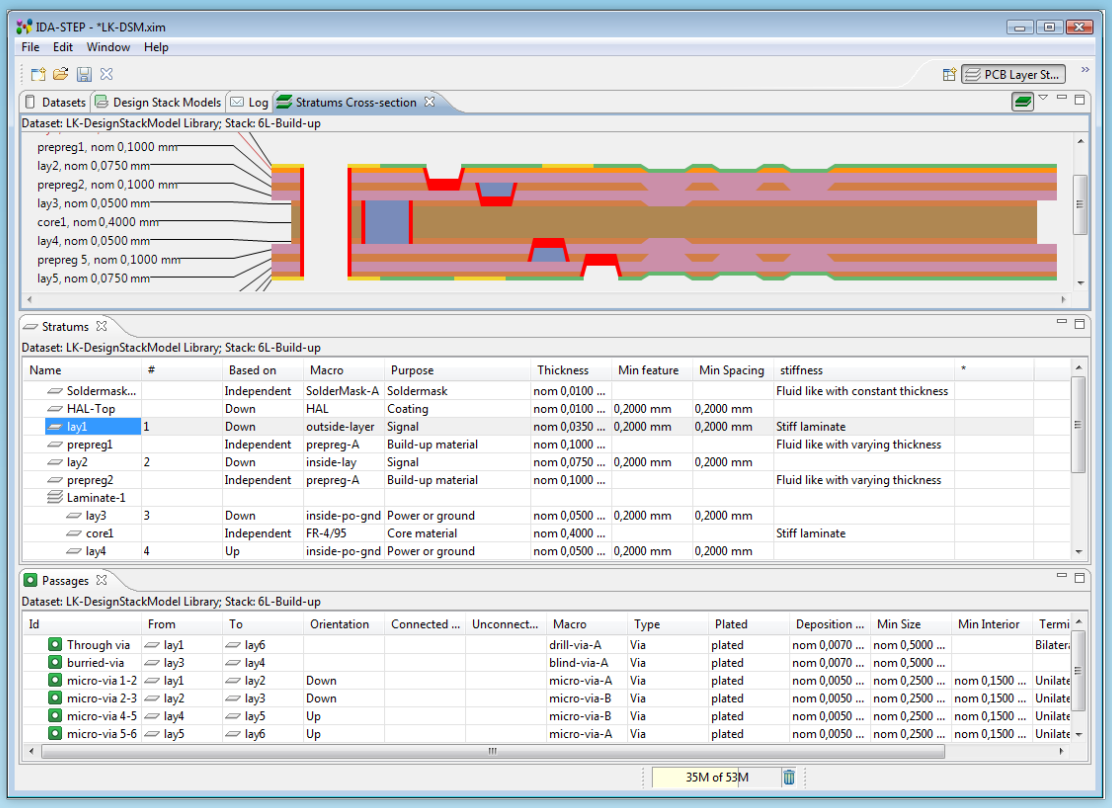
IDA-STEP v4

http://www.ida-step.net

PCB Layer Stack Editor

Successful coordination between PCB designer, fabricator and management



PCB technology: design, documentation, exchange, selection and review

- Conformance to **ISO 10303-210** and IPC-2226 on HDI
- **Layers**: Electrical and all others; laminates
- **Passages**: Vias, through holes, outline, cutouts
- Standard and user defined properties with tolerances
- Library of design layer stack families
- Regions with different layer stacks (flex-ridget)

LKSoftWare GmbH
http://www.lksoft.com
Steinweg 1 (Pilgerzell),
36093 Kuenzell, Germany
Tel.: +49 661 933933 0
Fax: +49 661 933933 2

PCB Layer Stack Editor

The PCB Layer Stack Editor is a tool to specify PCB technologies independent of particular PCB designs. It provides the following advantages:

- Efficient and successful coordination among PCB designer, fabricator and the intermediate procurement process
- Precise, formal and standardized information prevents communication errors
- Enables formal checking of the given PCB technology information for completeness and consistency

In addition the results of the PCB Layer Stack Editor can be used to complete the output of PCB-CAD systems, either by associating it with the delivered GERBER files or by linking it with native design data.

PCB Layer Stack data can be displayed by the free IDA-STEP v4 Viewer.

IDA-STEP v4

The PCB Layer Stack Editor is a component of IDA-STEP v4 and can be combined with other components. For example, using the PCB-CAD import components a PCB design can be integrated with technology data.

Standardized data exchange

Data can be exported to and imported from normative STEP AP210 files according to ISO 10303-210. Thus the data can also be used as input for simulation tools such as warpage, producibility and reliability.

Layers and other stratums

Purposes:

- Dielectric: core, build-up material
- Electrical: signal, power/ground, lands only
- Finish coating (lands)
- Core material, build-up material
- Paste mask, solder mask, solder paste
- Glue, glue-mask
- Silkscreen
- Others: generic, embedded passive (resistor or capacitor/dielectric)

Characteristics:

- Material identification
- Thickness (min, max, nom, plus/minus)
- Stiffness class: stiff, fluid-like, fluid like with constant thickness
- Min. and max. feature size
- Min. feature spacing
- User defined properties

Passages

Purposes:

- Via (for vertical connections only)
  - Through via
  - Blind via
  - Buried via
  - Micro via
- Through hole for wired components
- Others such as mounting holes
- Outline and cutout edge

Characteristics:

- Plating with material identification
- Filling with material identification
- Min. hole diameter
- Max. aspect ratio
- Orientation (up/down), needed for micro via
- Straight or tapered
- Annular ring for each layer
- Terminus condition (bond or removal)
- User defined properties

PCB Layer Library

PCB Layer Library is capable of storing similar PCB data to reuse it in multiple designs.

- Management of layer stack families sharing identical passages and stratums
- Creation, storage and reuse of macros - named passage and stratum technologies with related materials