

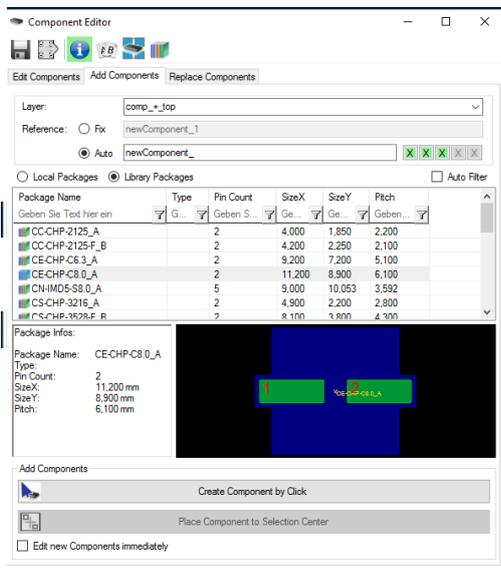
Release PCB-Investigator 8.0

Here is a short overview about the newest updates in version 8.0 of PCB-Investigator.

All Features can be tested in our free trial versions, have a look to our webpage here:
<http://www.pcb-investigator.com/en/download>

Nov 2016: New features PCB-Investigator 8.0.0:

- Component editor is connected with component library, now you can manage components and replace 3D elements simply.

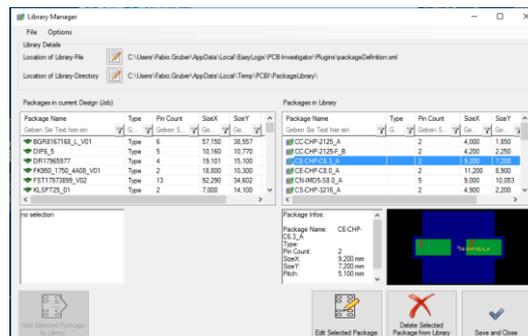


> Edit components in your design

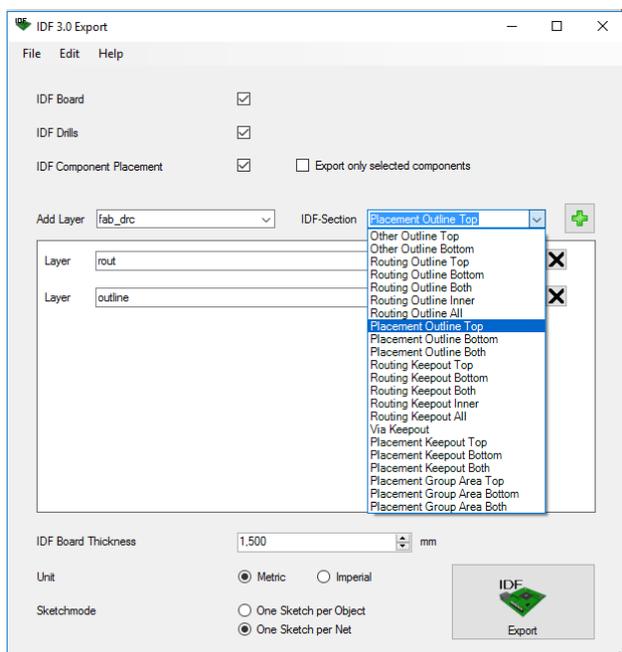
> Add new components with packages from a connected library

> Replace groups of components/packages by elements from your library e.g. with 3D models or with complete attribute sets.

> Create new package definitions e.g. by selecting the mask pads and modify the outline in an extra editor.



- Update of IDF, it's faster to export and import and now you can write much more details in output files.

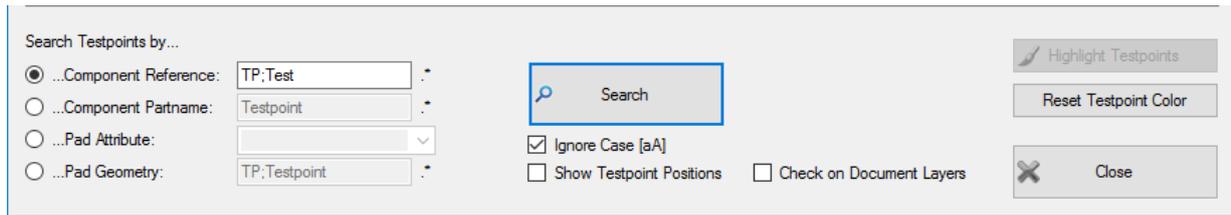


> Flexible options for each layer

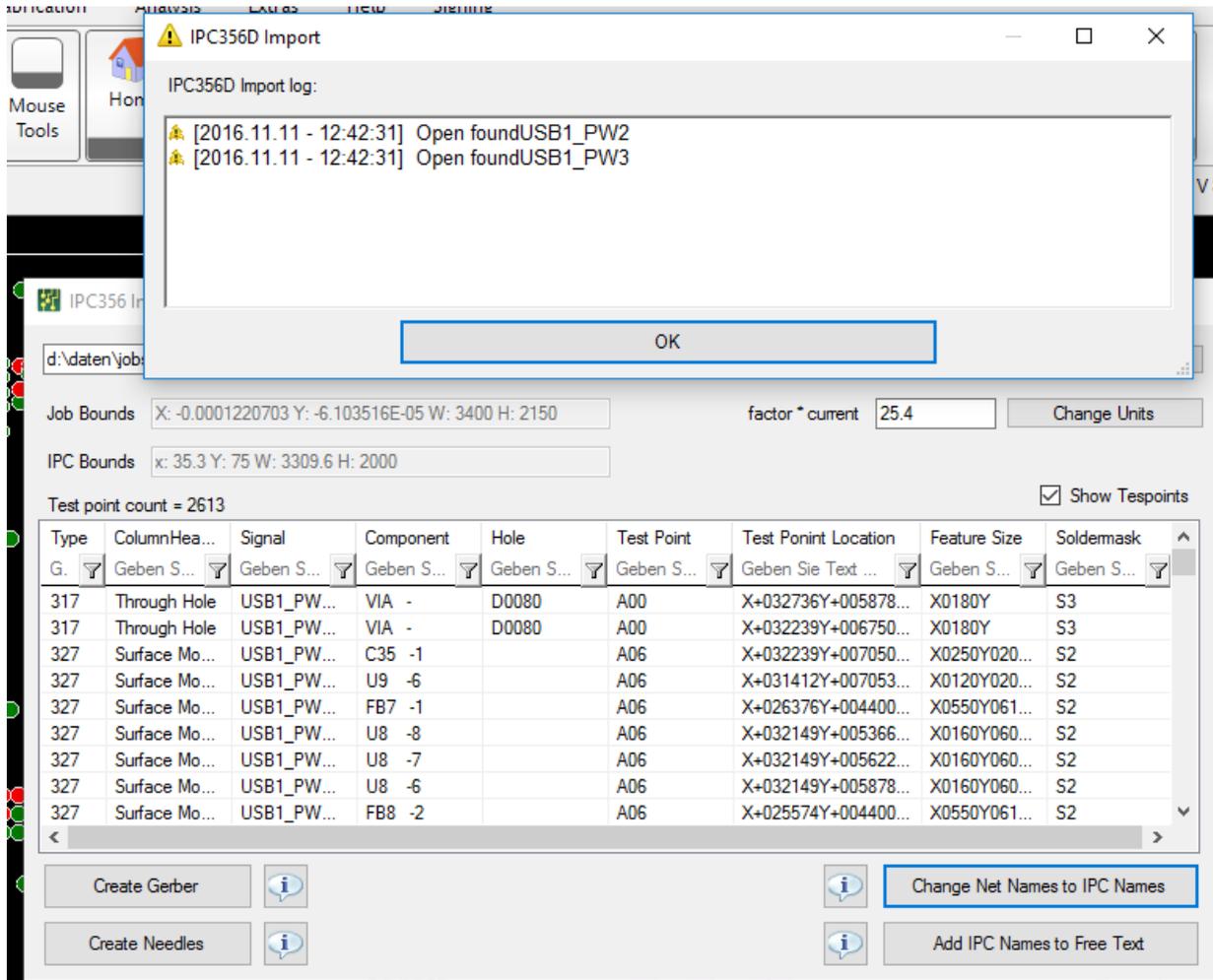
> Much easier to use than older IDF export

> Updated with all available IDF sections

- Add new options to find testpoints (also added some more information by importing GenCad testpoints).

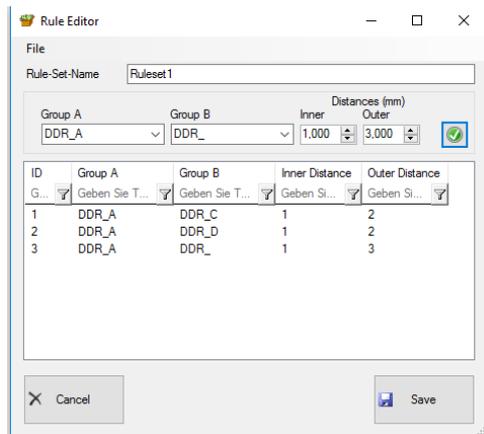


- Import of IPC356D netlist do short and open checks immediately.



- Using of component library while importing CSV component information or attributes. The new library concept of component editor is connected to the CSV component import, than you do not need extra Files with package definitions and can manage all packages in one library.

- Add many new options and tools to net group wizard (e.g. distance checks for groups with rule files).



> see rule editor in the net group distance calculator

> Import global rule files for company internal rule sets.

> Add net attributes to ODB++ data to save the net groups directly in the design.

- Compare mapping to set default values in graphically board compare.
Define company mapping files (e.g. you have older data where all layers are layer1, layer2,... and new designs where the layer named with top, inner1,... than you can use the mapping file to combine the layer1 with top and layer2 with inner1 ...
- Flatten step and repeat is faster than ever.
This option was very slow in the past, now it works in short time.

Interface updates:

- New IAttribute class added much more methods for handling of properties and (user) attributes.
- With IEDA_PRP class you have easy way to handle properties for all types of elements.
- IDrawingParameters are added to handle the image creating exactly.
- INote class for creating and showing your own note elements.

See <http://www.pcb-investigator.com/en/sdk-participate> for more details.