Integr8tor v2022.06

Fixed issues

Your continued feedback is important and appreciated. This version solves the following issues you have raised with our Customer Care department.

- Jobs with copper-balancing filling patterns had the potential to bog down PadMaker and could potentially cause the job to fail. PadMaker has been overhauled to no longer suffer from this.
- When an archive contained 2 drill files in which the same tool number was used with the same end diameter, and the zip file was submitted to Integr8tor using a .ttb, the manual assignment of different tool diameters to each of them in Drill Editor afterwards, was not carried forward correctly into the design's analysis. At the end of the process, both tools would show identical tool diameters. This issue has been addressed.
- Under circumstances, the plating information on routed slots could get lost during processing, leading to false netlist compare errors being reported at the end. The 2022.06 release corrects this software malfunction.
- The presence of parenthesis in the gerber file names inside an archive caused an "Unexpected error during Import" and the data would fail to read in. The Integr8tor input routines have been revised to now deal with this correctly.
- The third tab page in the Layer Structure Editor (above the Zoom out – Total – Measure – Flip Job buttons on the far right) had lost the ability to show the contents of the file that was clicked in the file list on the left. This regression has been undone and file contents are now showing correctly again.
- The presence of a '+' sign in the path to a job's work directory would prevent the Stackup tab page in the Layer Structure Editor and Drill Editor from displaying layer images correctly and crippled all associated viewing functions like zoom/pan/total/… along with it. The image viewers inside Layer Structure Editor and Drill Editor have been made more robust to handle this type of situation. This '+' typically appears in cases where the original archive file name contains a '+' (e.g. 33+411.zip) and the Integr8tor Work folder definition in Preferences is set up to include the %fileName% variable.
- When Integr8tor was uncertain about the correct polarity of a solder mask layer (orange background for the corresponding cell in the Polarity column in File List), it neglected to issue a todo for this. This release better warns the user about uncertain layer polarity issues by flagging them in a consistent and streamlined way.
- The layer polarity recognition algorithm has been optimized to come out with higher first-pass hit rates, avoiding time-consuming re-analysis and saving out on workflow capacity.
- The combination in an archive of both a .DRR file listing rounded tool sizes, and Excellon drill/slot files with unrounded tool sizes in the header, could lead to a stalemate causing Integr8tor to say it was unable to find any tool sizes. This code deadlock has been removed and the system will now prefer the sizes in the Excellon headers over the ones in the .DRR.
- The tool size recognition for drill files with tool diameters in mil could produce diameters that were too small. The improved evaluation of drill sizes in this 2022.06 release puts this issue to rest.
- When an archive contained the same drill data in both Excellon and Gerber format, the Excellon was initially demoted for the benefit of the Gerber variant, but later on in the process, the Gerber could then be qualified as a drill drawing, rather than as a drill. This mis-qualification has been corrected and the Gerber variant will now correctly end up as drill.

Ucamco nv
Bijenstraat 19, 9051 Gent, Belgium
Tel: +32 9 216 99 00 – Fax: +32 9 216 99 12
www.ucamco.com - info@ucamco.com