



# eCADSTAR™

## DATASHEET

# GERBER IMPORT

A modest-sounding capability with nothing to be modest about. Importing Gerber data in eCADSTAR can get your PCBs to fabrication more quickly and reliably.

Gerber data is still the de facto format for PCB fabrication. It describes artwork, including conductors, solder mask and drill data. PCB fabricators take it, tweak it, panelize with it and turn it into PCBs. It's clear what will happen when you export Gerber from eCADSTAR, but what can you do when you import Gerber?

Some very useful things indeed. Things that save you lots of time and help you avoid errors.

### TOP BENEFITS WHEN YOU IMPORT GERBER DATA

- Make sure what's getting fabricated matches your intention.
- Re-use legacy PCB data, even if photo data is all you've got.
- Use reference design data from any source that supports Gerber.
- Transform photo data into real, editable eCADSTAR PCB data.

There's much more than there's room to mention here that helps you get even more from importing Gerber data in eCADSTAR PCB Editor.

Why not try it for yourself?

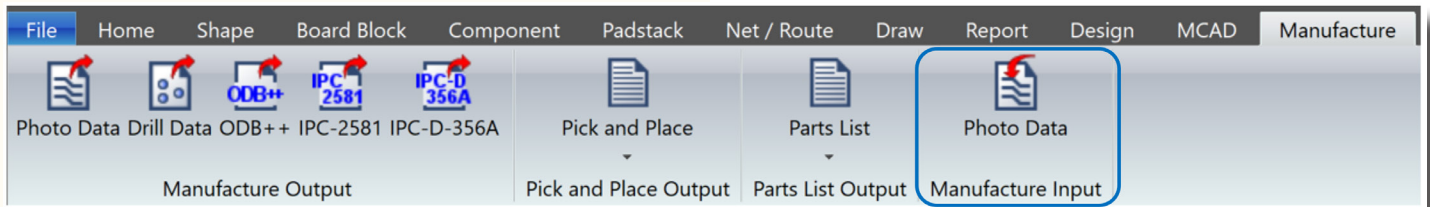
[www.ecadstar.com](http://www.ecadstar.com)

©Zuken Inc. All rights reserved.

Save a tree. Please don't print this datasheet unless you really need to.

## GERBER IMPORT IN PCB EDITOR

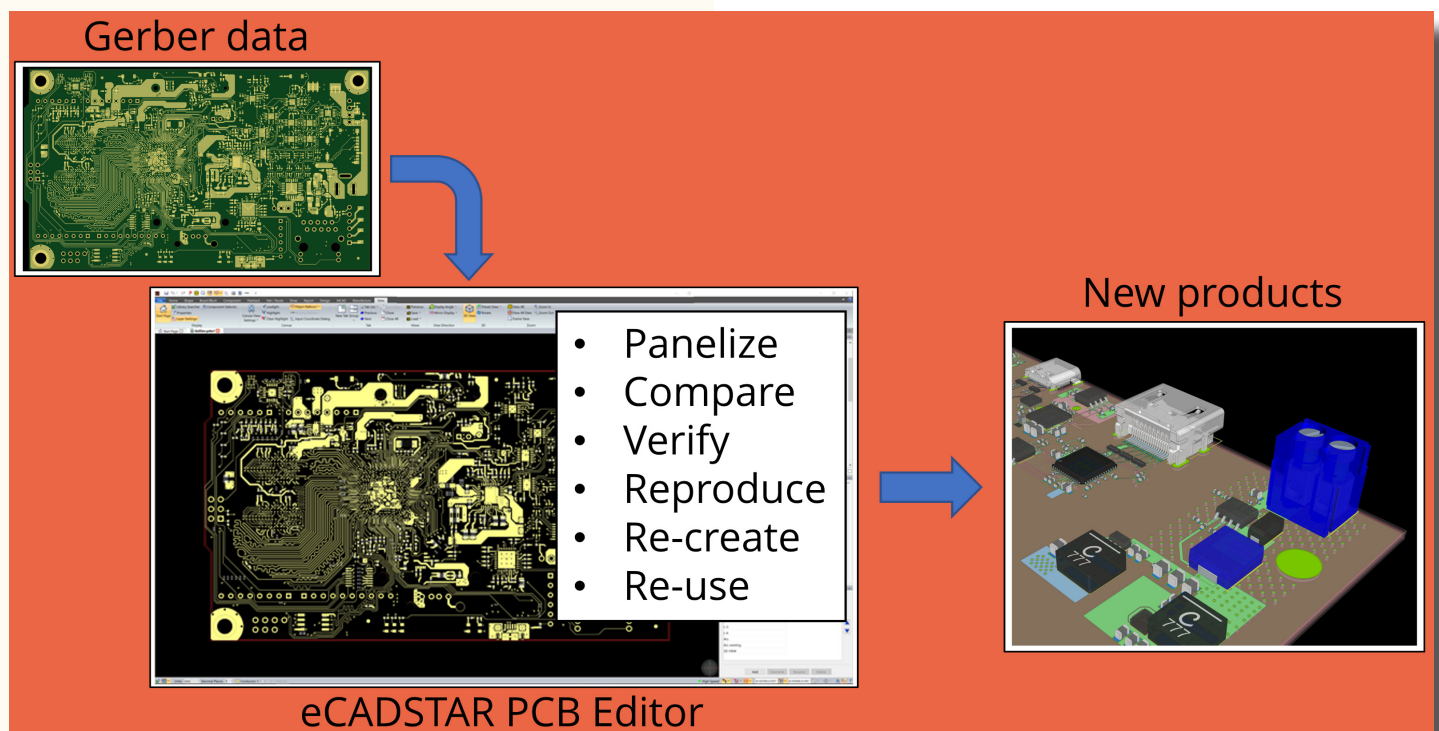
When you import Gerber data you'll be presented with a rich set of options. You'll find it on the Manufacture tab.



**FIGURE 1:** eCADSTAR ribbon with Gerber import capability

## WHY IMPORT GERBER?

Here are some of the reasons.

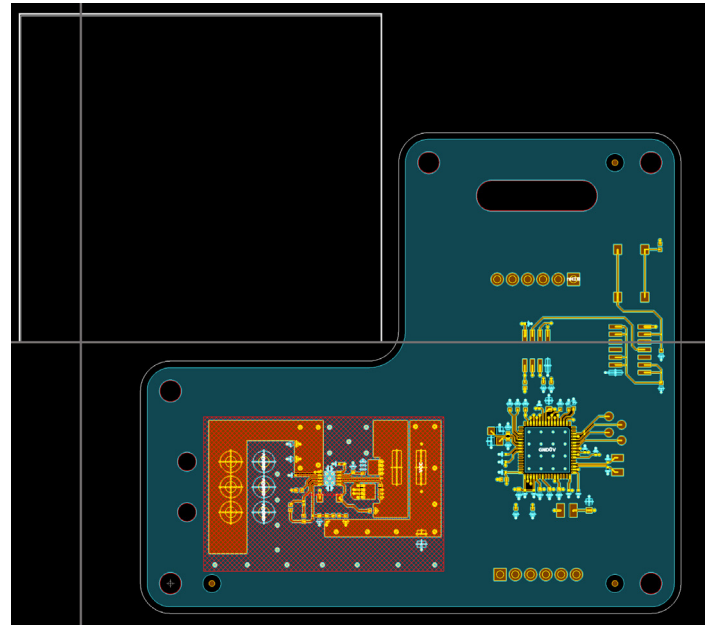


**FIGURE 2:** Useful things you can do when you import Gerber

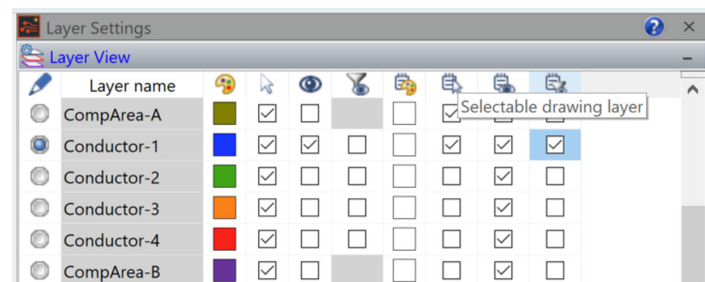
**Compare and verify** imported artworks, including versions “tweaked” by fabricators, comparing with what’s on your PCBs and making sure your intent hasn’t been lost. eCADSTAR drawing layers exist alongside the physical layers on your PCB. You can import Gerber data on to a drawing layer and see it together with a physical layer. You can adjust transparency to see differences easily. Review Gerber data without the need of an external viewer.

**Reproduce, re-create and re-use** from photo data, layer-by-layer, reuniting artworks with lost nets and connections. You can duplicate artwork data that you’ve imported on to a drawing layer directly on to a physical PCB layer. The artwork transforms into real PCB tracks and shapes. At that stage, the tracks don’t know which nets they belong to. You can add those net labels yourself or you can get eCADSTAR PCB to do it for you. If you drop a component into position, net labels are assigned automatically, saving a lot of work. eCADSTAR detects connections without needing identical pin, pad and track reference points.

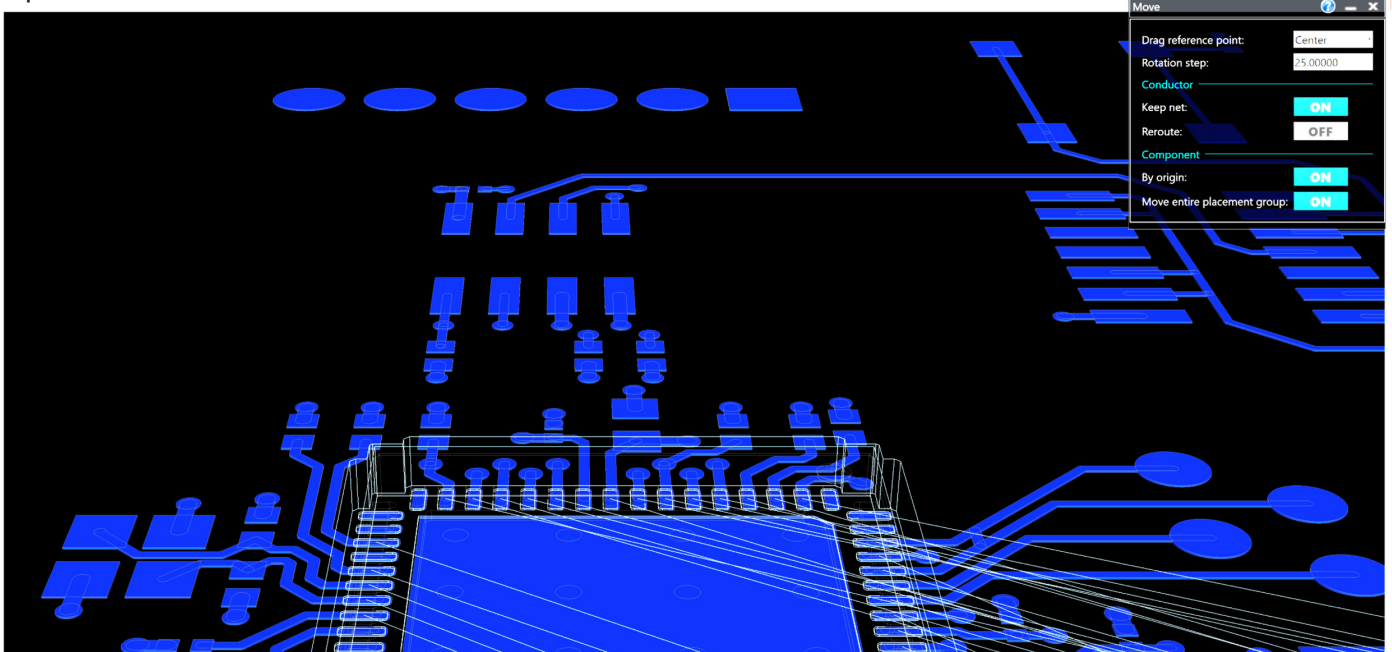
**Panelize** your PCB design, importing Gerber data and arranging your PCBs for maximum economy. They needn’t even be for the same product.



**FIGURE 3:** Dropping imported artwork into place on a panel



**FIGURE 4:** Drawing layers exist in parallel with physical layers



**FIGURE 5:** When you drop a component into place, net labels are automatically assigned to imported tracks