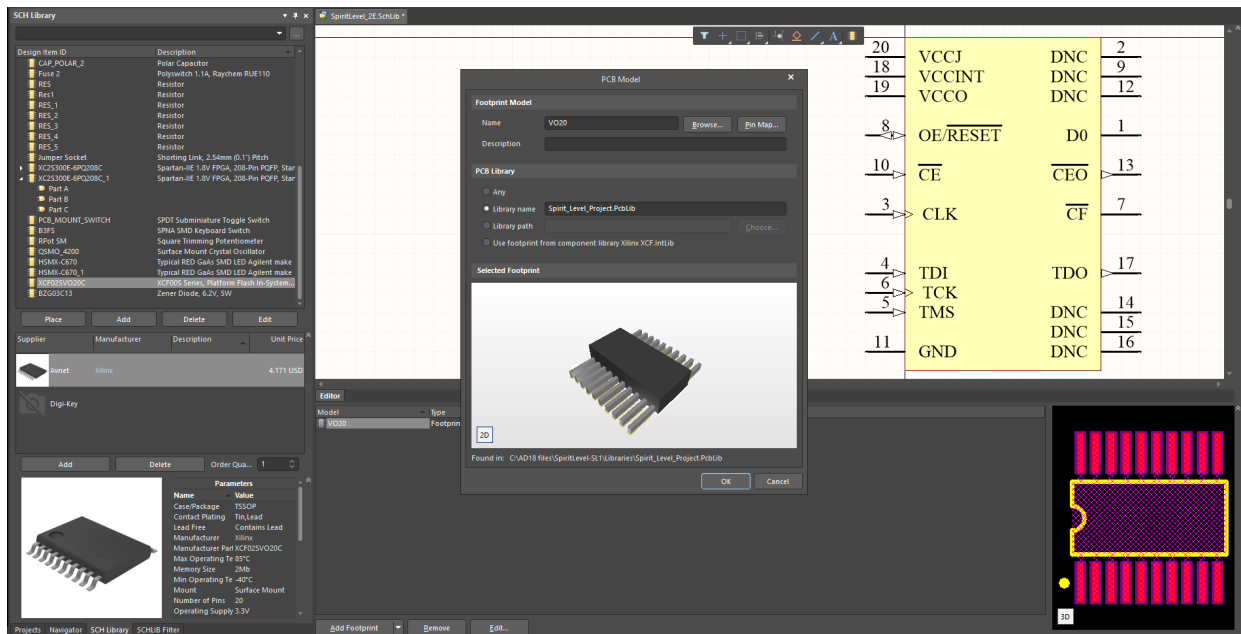# Altium Designer® 18

FEATURES AND BENEFITS

- Centrally manage your component libraries so they are always up-to-date

- Easily make data-driven, intelligent component decisions to save time and money

- Minimize uncertainties with access to component availability, pricing, and lifecycle state

- Save time by automating the creation of IPC-compliant component footprints

COMPONENT MANAGEMENT

We all know that a simple process of selecting a part for a design can spiral into a lengthy agony of undoings — not knowing, for instance, whether the part selected will enter end-of-life before the project is even complete. Or sometimes last-minute supply issues derail design timelines and increase costs. Altium Designer® component management ensures that these kinds of scenarios are minimized by leveraging complete control over the component selection process. Integrated supply chain information combined with specified alternate parts directly in the bill of materials (BOM) minimize supply issues, selecting unapproved parts, or designing with components with unknown lifecycle states.

Unified Component Model

Altium Designer uses different library types (Schematic, PCB, Database, etc.) to define different aspects of a component (Symbol, Footprint, etc.) unified by a component library. Unified component models combine all defining information into a single, placeable design part. Maintained libraries exist in the same ecosystem as the design environment, allowing direct placement on a schematic and PCB layout.

You can link parts with real-time pricing and availability data from personal parts suppliers and over 100 Altium verified suppliers. You can leverage price, availability, and real-time supplier information to meet your design goals. Utilizing the information reduces risks of last minute costs due to component supply chain issues, such as long lead times or end-of-life (EOL). With Altium Designer component management, you can keep components organized, gather usage statistics, and update components to the latest revision with the click of a button.

Unified Component Model Management

Real-Time BOM Management

The BOM is a list with the necessary parts used in a design for manufacturing. ACTIVEBOM® provides you with automation by supplying part information such as availability and price from selected suppliers. Altium Designer ACTIVEBOM allows specifying pin-compatible backup part choices directly in the BOM referred to as alternative part choice. Having pin-compatible backup part choices nearly eliminates supply chain issue risks for manufacturing. In turn, you can design while taking into account potential manufacturing blowouts, reducing time to market and minimizing unexpected costs and design changes.

ActiveBOM - Real-Time Cost Estimation & Part Availability

IPC-Compliant Footprint Wizards

Part creation is one of the most time-consuming tasks a designer has to do. Not only do you need to copy and paste the data into the component properties but you need to accurately draw each footprint before you can start your layout. With the IPC-compliant Footprint Wizard, you no longer need to spend your time drawing and redrawing each component footprint. The wizard creates IPC-compliant component footprints, and also 3D models. Rather than working from footprint dimensions, the IPC-compliant Footprint Wizard uses dimensional information from the component itself in accordance with the standards released by the IPC.

Rather than requiring you to enter the properties of the pads and tracks that are used to define the footprint, the Footprint Wizard takes the actual component dimensions as its inputs. Then, based on the formulas developed for the IPC-7351 standard, the Footprint Wizard generates the footprint using standard Altium Designer objects, such as pads and tracks, allowing you to edit and modify.
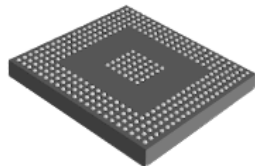

IPC-Compliant footprint Wizard