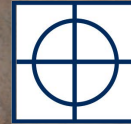




**SIEMENS**

*Ingenuity for life*



**GEOMETRIC  
SOLUTIONS**

# Teamcenter integration for Cadence Allegro products

backbone. Teamcenter's Allegro integration provides a comprehensive solution for the entire electronics products lifecycle that extends from initial inception through creation, analysis, manufacturing, service and end-of-life disposition.

## Providing a single source of product and process knowledge

Teamcenter's Allegro integration enables users to access, manage and archive PCB design data in a single secure location. The integration lets users open and save native design files, access approved components, populate electrical component BOMs and share fabrication and assembly data – as well as create and manage derived files directly in Teamcenter.

Teamcenter menus, embedded in the Allegro user interface, allow the user to automatically login into Teamcenter

## Integrating Allegro PCB design into your Teamcenter PLM environment

### Benefits

- Manages entire PCB product lifecycle
- Provides a single source of product and process data
- Facilitates today's collaboration and concurrent engineering initiatives
- Leverages requirements management

### Business challenges

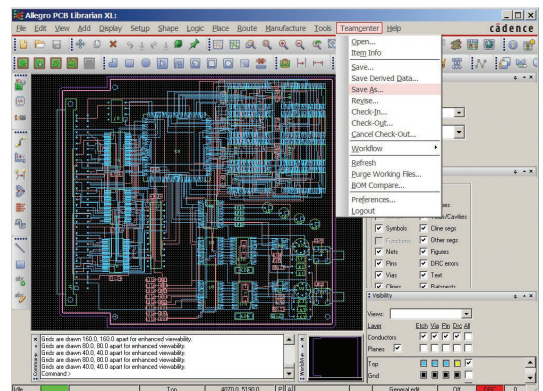
- Product development organizations use multiple ECAD and MCAD design systems
- Product teams need to access and share all of the diverse information that they create
- Design teams are located in multiple sites or dispersed across a multivendor supply chain
- Product change and approval processes take too long

### Summary

Teamcenter® software's integration for Cadence Allegro PCB design enables companies to accelerate time-to-market and reduce development cost by allowing users to capture their schematic, PCB, bill of material (BOM), fabrication and assembly data in Teamcenter – the world's most widely used product lifecycle management (PLM) solution.

### Managing the electronics product lifecycle

The Teamcenter integration for Cadence Allegro PCB Design enables users to capture PCB design data (schematic, physical layout, BOM, fabrication, assembly and visualization data) created under Allegro and store/manage this information within Teamcenter – Siemens PLM Software's digital PLM



Teamcenter menu is embedded in the schematic and layout tools so designers never have to leave their native environment.

# Teamcenter integration for Cadence Allegro products

## Features

- Create, edit and store Cadence Allegro objects (schematic, components, PCB, BOM, etc.) as items in Teamcenter
- Leverage Teamcenter menus embedded in the Cadence Allegro user interface to facilitate ease of use
- Manage designer's selection and use of approved components
- Establish relationships between Teamcenter objects and engineering BOMs
- Share PCB design data across engineering domains through open interchange formats
- Associate and trace product requirements to PCB designs
- Create new product revisions or version updates for work in progress
- View, annotate and mark up design data in a collaborative environment
- Establish security and data access control

and open, save, check-in and check-out design data. The integration assures design teams their ECAD data is accurately captured and consistently managed in the Teamcenter environment so it can be kept in-sync with other product definition data.

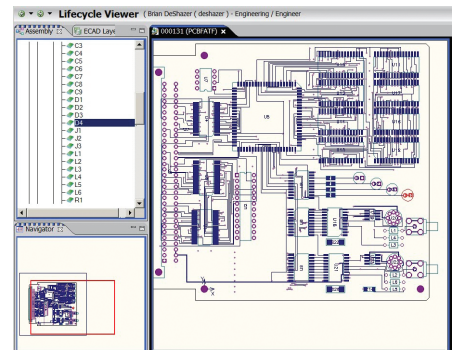
On an enterprise level, the integration allows widely dispersed PCB design teams to manage released design data, collaborate and execute design changes across the entire product lifecycle, thereby minimizing change-related rework.

## Facilitating collaboration and concurrent engineering

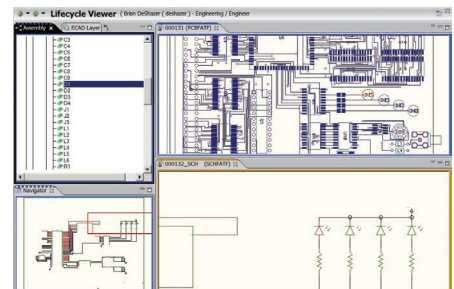
To facilitate the flow of accurate design data across multiple domains, the Teamcenter integration enables collaborative design by leveraging IDF and IDX (EDMD) design data exchange formats. The formats enable the sharing of information relating to board outlines, component placements, keep-out areas and other placement restrictions. Managed in Teamcenter the IDX format supports the ability to pass incremental design data, as well as allow both the ECAD and MCAD designers to accept or reject changes, and incorporate change notes or comments into the information being shared.

Electrical engineers can pass this information as 2.5D/3D elements to mechanical engineers to simulate and analyze various conditions, including interferences, thermal, vibration, shock, dust and humidity. Sharing data for this type of cross-domain analysis helps improve quality and increase product reliability.

To quickly diagnose and understand potential manufacturing errors, users can employ optional design-for-assembly analysis tools and powerful ECAD viewer technology. These capabilities allow users to investigate and identify potential issues early in the design process, thereby eliminating unnecessary scrap and rework.



Design teams and suppliers can easily visualize and mark-up design issues.



Electrical engineers and PCB designer can cross-probe between schematic and physical layout to zero in on objects of interest.

The ECAD viewer's graphical navigation features enable design teams and suppliers to interactively view, cross-probe and annotate schematic and PCB layout data without the use of an expensive authoring tool. Many frequently used annotations are automatically translated and displayed using the language specified by the user's system.

## Complete requirements management and traceability

Teamcenter's Allegro integration enables users to optionally leverage Teamcenter's powerful requirements management capabilities. PCB hardware and software functions can be associated with specific design requirements, providing complete requirements traceability throughout the entire PCB lifecycle.

### Supported objects

The integration with Cadence Allegro captures:

- Circuit card assembly (CCA) information
- Components on a CCA
- Schematics
- Allegro design data
- Secondary data (fabrication and assembly)
- MCAD and analysis interchange files
- Allegro layout file(s) in native tool format
- Schematic file(s) in native tool format
- Neutral format files for PCB and schematic visualization

### Supported functions

- Open, save, check-in and check-out objects to/from Teamcenter
- Establish security and data access control policies in Teamcenter
- Extract components and attribute information
- Generate bill of material (BOM)
- Place Allegro objects under enterprise-wide revision control
- Manage Allegro objects in structured workflows

### Options

- Facilitate enterprise-wide ECAD library management
- Manage Allegro objects in formal change processes
- Link Allegro objects to product/project requirements
- Leverage ECAD viewer and markup capabilities with suppliers
- Cross-probe between schematic and PCB layout
- Analyze layouts against assembly rules

Siemens PLM Software  
[www.siemens.com/plm](http://www.siemens.com/plm)

Americas +1 314 264 8499  
 Europe +44 (0) 1276 413200  
 Asia-Pacific +852 2230 3308



844-GEO-SUPT  
[support@geoplms.com](mailto:support@geoplms.com)  
[geoplms.com](http://geoplms.com)

© 2017 Siemens Product Lifecycle Management Software Inc. Siemens and the Siemens logo are registered trademarks of Siemens AG. D-Cubed, Femap, Fibersim, Geolus, GO PLM, I-deas, JT, NX, Parasolid, Solid Edge, Syncrofit, Teamcenter and Tecnomatix are trademarks or registered trademarks of Siemens Product Lifecycle Management Software Inc. or its subsidiaries in the United States and in other countries. All other logos, trademarks, registered trademarks or service marks used herein are the property of their respective holders.  
 12830-A8 1/17 B