

Customer Onboarding Guide



Welcome to Altium

We're excited to have you join the Altium family! The Team Up, Trade Up bundle is designed to give your team the tools and collaboration capabilities they need to modernize your electronics design workflow. After completing the steps in this onboarding guide, you'll be able to:

- Provide access to best-in-class design technology in Altium Designer for up to 10 team members with [Pro Subscription benefits](#).
- Enable seamless global collaboration across ECAD, MCAD, and Supply Chain specialists with Altium 365's cloud-based applications.
- Align Project Management, Design, and Procurement with accurate, real-time data using Jira integration and unlimited cloud storage.

How to Get Help

Your bundle provides access to real-time support, user communities, comprehensive product documentation, training videos, and a knowledge base of targeted solutions for common user experiences.

Your support resources

- [Visit Altium Support Home](#)
- [Join Our Weekly «Ask an Expert» Sessions](#)

Getting Started

We recommend following the steps below to get started with Altium Designer and Altium 365's cloud-based applications that are included in your bundle. Each step builds on the one prior to ensure a smooth onboarding experience for you and your team members.

You can also explore these two Getting Started Guides to supplement your learning:

- [Altium Designer Getting Started Guide](#)
- [Altium 365 Getting Started Guide](#)

Customer Onboarding Guide



Step 1: Account Activation & Downloads

In this step we'll take care of the basics - creating your Altium Account, installing Altium Designer, and activating your team's Altium 365 Workspace.

ONBOARDING OBJECTIVES	RESOURCE
Create your Altium Account	Register here
Download and install Altium Designer	
Note: You'll sign in to the Altium Account you created above during the install process.	Download here
Active your team's Altium 365 Workspace	Activate here

For additional details about Altium 365 Workspace activation and management, please see [this documentation article](#).

Step 2: Environment & Access Setup

Now that you have access to Altium Designer and Altium 365, you can focus on inviting your team members and familiarizing yourself with your new desktop and web interfaces.

ONBOARDING OBJECTIVES	RESOURCE
Invite team members to your Altium 365 Workspace and learn how to manage content access	Documentation: Managing Workspace Membership
Get familiar with your Altium Designer environment & system preferences	Documentation: Elements of the Environment Documentation: Accessing, Defining & Managing System Preferences
Get familiar with your Altium 365 Workspace	Documentation: Altium 365 Platform Interface
Define and manage lifecycle definitions for your Workspace	Documentation: Lifecycle Management
Configure part sources for your Workspace	Documentation: Part Source Configuration
Configure global operation permissions for your Workspace	Documentation: Setting Global Operation Permissions for a Workspace
Learn how to use version control in Altium Designer & Altium 365	Documentation: Using Version Control

Step 3: Importing Libraries & Using Cloud Libraries

Your team might have existing component libraries that need to be imported to Altium. This step covers the import process and how to create components in your new [Altium 365 Cloud Library](#).

ONBOARDING OBJECTIVES	RESOURCE
Import existing libraries to your Workspace	Documentation: Importing Existing Libraries to Your Connected Workspace
Learn how to create new components in Altium Designer	Getting Started Guide: Creating a New Component

Customer Onboarding Guide



Step 4: Creating & Managing Projects

Now that your team members have been invited and your component libraries are imported, you can start working on your projects and collaborating with commenting and tasks.

ONBOARDING OBJECTIVES	RESOURCE
Learn how to create a new project or upload existing projects to your Workspace	Documentation: Workspace Projects
Learn how to create Project Templates	Documentation: Creating a Project Template
Learn how to create Schematic Templates	Documentation: Creating a Schematic Template
Learn how to share a design with team members and modify existing access rights	Documentation: Sharing a Design Documentation: Modifying Access Rights for Existing People
Learn how to create and manage Comments in your project documents	Documentation: Document Commenting
Learn how to create and manage Tasks in your Workspace	Documentation: Working with Tasks
Learn how to use Project History to see a complete view of a project's development journey	Documentation: Project History

Step 5: Improving Cross-Team Collaboration

In this final step, you'll learn how to extend your electronics design workflow to collaborate with MCAD, procurement/supply chain, project management, and simulation specialists.

ONBOARDING OBJECTIVES	RESOURCE
Learn how to collaborate with mechanical engineers with MCAD CoDesigner	Getting Started Guide: Introduction to Altium MCAD CoDesigner
Learn how to collaborate with Ansys simulation specialists with Ansys CoDesigner	Documentation: Ansys CoDesigner
Learn how to enhance your BOM management workflow with BOM Portal	Documentation: BOM Portal
Learn how to enable bi-directional synchronization between Altium 365 and Jira	Documentation: Jira Application

Closing Thoughts

After completing all of the steps in this onboarding guide, your team should be well-equipped to design electronics using a modern workflow in Altium! We hope this journey continues to be transformative for your team and that you discover new ways to save valuable engineering time.

If you need any help at any time, please contact your [Altium Support team](#).

All the best,

Your Altium Team