

# Customer Onboarding Guide



## Welcome to Altium

This onboarding guide is designed to give your team the tools and collaboration capabilities you need to modernize your electronics design workflow. After completing the steps in this onboarding guide, you will 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](#)
- [Altium Learning Hub](#)
- [Altium 365 Training](#)

## 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 will cover creating your Altium Account, installing Altium Designer, and activating your team's Altium 365 Workspace.



ONBOARDING OBJECTIVES	RESOURCE
Create your Altium Account	 <a href="#">Register here</a>
Download and install Altium Designer* Note: You'll sign in to the Altium Account you created above during the install process.	 <a href="#">Download here</a>
Active your team's Altium 365 Workspace	 <a href="#">Activate here</a>

## Step 2: Environment & Access Setup

Now that you have access to Altium Designer and Altium 365, you can invite your team members and get familiar with both the desktop and web interfaces. (Also covers importing existing component libraries.)

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

## Step 3: Importing Libraries & Using Cloud Libraries








ONBOARDING OBJECTIVES	RESOURCES
Import existing libraries to your Workspace	 <a href="#">Documentation: Importing Existing Libraries to Your Connected Workspace</a>
Learn how to create new components in Altium Designer	 <a href="#">Getting Started Guide: Creating a New Component</a>

# Customer Onboarding Guide







## Step 4: Creating & Managing Projects

Once your team members have been invited and component libraries have been imported, you can begin working on your projects with commenting and task management.

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

## Step 5: Improving Cross-Team Collaboration

Extend your electronics design workflow to collaborate with MCAD, procurement/supply chain, project management, and simulation specialists.

ONBOARDING OBJECTIVES	RESOURCES
Learn how to collaborate with mechanical engineers using MCAD CoDesigner	 <a href="#">Getting Started Guide: Introduction to Altium MCAD CoDesigner</a>
Learn how to collaborate with Ansys simulation specialists with Ansys CoDesigner	 <a href="#">Documentation: Ansys CoDesigner</a>
Learn how to enhance your BOM management workflow with BOM Portal	 <a href="#">Documentation: BOM Portal</a>
Learn how to enable bi-directional synchronization between Altium 365 and Jira	 <a href="#">Documentation: Jira Application</a>

## 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](#).