SOLIDWORKS

2026

BLOG

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Learning Tutorials

A Note to Our Readers

2026

This blog has been created using a combination of artificial intelligence tools and human review to help deliver clear, structured, and up-to-date learning content.

All technical topics, examples, and workflows are curated to support learning and skill development. While every effort is made to ensure accuracy and clarity, readers are encouraged to validate concepts through hands-on practice and documentation. Our goal is to make learning more accessible, efficient, and practical for everyone.

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— CADIN360 Team

AVOIDING ACCIDENTAL DELETIONS IN SOLIDWORKS

• LEARN • • APPLY • • GROW •

Introduction

Accidental deletion of files or parts in SolidWorks can cause significant delays, data loss, and frustration. As one of the most widely used CAD software, SolidWorks offers powerful modeling tools, but managing files correctly is essential to prevent costly mistakes. Avoiding accidental deletions in SolidWorks isn't just about careful work—it's about adopting proactive strategies, best practices, and understanding the software's features to safeguard your designs. Whether you're a beginner or an experienced user, this guide provides practical, step-by-step advice on how to protect your work, manage file versions, and ensure your projects are safe from unintended deletion.

Understanding the Causes of Accidental Deletion in SolidWorks

Before diving into prevention techniques, it's important to understand why accidental deletions happen. Common causes include:

- User error due to rushing or distraction
- Misuse of delete commands
- Deleting the wrong component or feature during complex assemblies
- Lack of proper version control
- Software glitches or file corruption
- Insufficient backups

By recognizing these causes, you can better adapt your workflow to avoid them.

Best Practices to Prevent Accidental Deletions

Implementing the right practices can dramatically reduce the risk. Here are the most effective strategies:

1. Use the SolidWorks Recycle Bin (if applicable)

- Although SolidWorks itself doesn’t have a dedicated recycle bin like Windows, it integrates with Windows Explorer.
- Always delete files from within SolidWorks or the associated file folder, and verify before deleting.
- Consider the Windows Recycle Bin as a safety net for deleted files.

2. Maintain Regular Backups and Version Control

- Save incremental versions of your files frequently.
- Use "Save As" with different filenames or version numbers (e.g., project*v1.sldprt, project*v2.sldprt).
- Utilize SolidWorks PDM (Product Data Management) systems for automated version control.
- Keep backups on an external drive or cloud storage for disaster recovery.

3. Enable SolidWorks AutoSave and Recovery Options

- Go to Options > Save, and enable AutoSave to automatically create backups at regular intervals.
- Adjust the AutoSave frequency depending on your work intensity.
- Use SolidWorks’ built-in file recovery features if the software crashes unexpectedly.

4. Lock Files and Parts

- Lock components, features, or assemblies using the "Lock" feature or configurations to prevent accidental modifications.
- Use configuration management to create stable versions that aren’t altered unintentionally.

5. Use the Undo and Ctrl+Z Feature

- Regularly use the Undo command (Ctrl+Z) immediately after making a mistake.
- Keep in mind, Undo is limited to the current session. Save frequently.

6. Mitigate Risks in Assembly Work

- When working on complex assemblies, suppress unnecessary components to reduce the risk of accidental deletion.
- Use component references and references to ensure components are correctly linked.
- Be cautious with delete operations—double-check before confirming.

7. Customize Toolbar and Shortcut Settings

- Remove or disable delete buttons from quick access toolbars for sensitive parts.
- Create custom shortcut keys to perform safe actions, reducing accidental deletions.

Step-by-Step: Safeguarding Your Files in SolidWorks

Here’s a practical workflow to prevent accidental deletion:

1. Set Up Proper File Management

- Organize your project folders logically and clearly.
- Save files with descriptive names and versioning.

2. Enable AutoSave and Backup Options

- Go to Tools > Options > System Options > Backup/Recover.
- Turn on AutoSave, and set the frequency (e.g., every 10 minutes).
- Specify backup locations.

3. Use Save As for Major Changes and Versioning

- After significant modifications, employ Save As to create a new version.
- Annotate file names to reflect versions and dates.

4. Activate Lock Features for Critical Parts

- Right-click on components and select "Lock" or manage via configurations.
- This prevents accidental edits or deletions.

5. Practice Undo and Confirmation

- Use Ctrl+Z immediately after unintended actions.
- When deleting, always double-check the selection and confirm prompts.

6. Implement PDM for Larger Teams

- Use SolidWorks PDM to control file access permissions and track changes.
- Set permissions to read-only for users not authorized to delete files.

Common mistakes to avoid when trying to prevent deletions

- Relying solely on the Windows Recycle Bin for file recovery—never assume deletion is recoverable without backups.
- Deleting files directly from the Windows desktop instead of within SolidWorks or project folders.
- Forgetting to save incremental versions during long modeling sessions.
- Disabling AutoSave or neglecting to back up files regularly.

- Not using PDM or version control systems in team environments.

Pro Tips for Advanced Users

- Create custom macros that prompt confirmation before deleting files or features.
- Use SolidWorks configurations to save different design states, enabling easy rollback.
- Take advantage of "Rollback" features within the Surface and FeatureManager design tree to revert features instead of deleting them.
- Enable notifications for file modifications when collaborating with teams.

Comparing File Recovery Methods in SolidWorks

Method	Effectiveness	Best For	Limitations
Undo (Ctrl+Z)	Quick	Recent accidental actions	Only during current session

AutoSave / AutoRecovery	Moderate	Software crashes or sudden closes	May not catch recent changes
File Backup / Versioning	High	Major mistakes or deletions	Requires prior setup
PDM System	Very high	Large teams with multiple users	Cost and setup required

Using multiple layers of protection enhances your chances of avoiding accidental deletions.

Conclusion

Avoiding accidental deletions in SolidWorks requires a proactive approach combining good file management, proper use of software features, and team collaboration tools. By implementing best practices such as regular backups, leveraging AutoSave, locking critical parts, and maintaining disciplined workflows, you can safeguard your work and minimize risks. Remember, prevention is always better than recovery. Ensuring your SolidWorks environment is optimized for data protection helps maintain productivity, prevents data loss, and keeps your projects on track.

FAQ

1. How can I restore a deleted part in SolidWorks?

Ans: If you haven't saved or emptied the recycle bin, restore the file from Backup, AutoSave, or version control.

2. What is the best way to prevent deleting the wrong component in an assembly?

Ans: Use component references and suppress unused components to avoid accidental deletion and ensure proper control.

3. How does SolidWorks PDM help prevent data loss?

Ans: PDM manages file permissions, tracks revision history, and controls access, reducing accidental deletions.

4. Can I recover a file if I accidentally deleted it from Windows Explorer?

Ans: Yes, if it is in the Windows Recycle Bin, you can restore it; otherwise, use backup or data recovery software.

5. What are some effective ways to manage versions of SolidWorks files?

Ans: Use Save As with version numbers, external backup systems, or PDM to maintain organized version control.

6. Is there a way to lock features within SolidWorks to prevent deletion?

Ans: Yes, you can lock features or use configurations to prevent unintentional modifications or deletions.

7. How can I ensure continuous data safety during extensive modeling phases?

Ans: Enable AutoSave, maintain regular backups, and use PDM for version control throughout the project.

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2026

CADIN360 Learning Tutorials is an educational platform focused on practical CAD, CAM, and CAE learning.

The platform provides clear, industry-oriented tutorials, design workflows, and real-world insights using tools such as SolidWorks.

CADIN360 is created to help learners, students, and professionals build strong fundamentals and practical design skills in modern CAD workflows.

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Learning Tutorials

Practice What You've Learned

2026

You've just completed this blog and learned important concepts in SolidWorks.

To help you practice and apply what you've learned, the next pages include a sample from our SolidWorks book .This sample contains practice exercises and real-world practice tasks designed to strengthen your skills.

What you'll find next:

- ✓ Practice exercises from the book
- ✓ A brief overview of the complete book
- ✓ Options to explore or request the full sample

Your hands-on SolidWorks practice starts next.

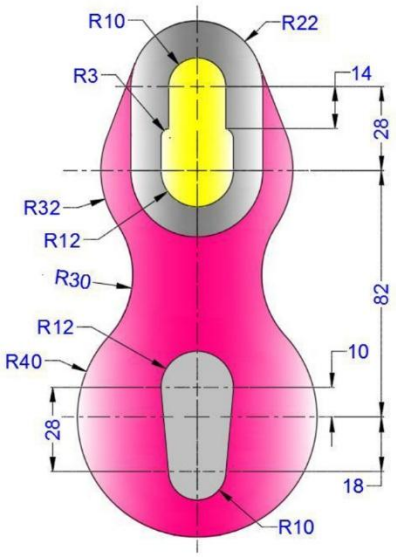
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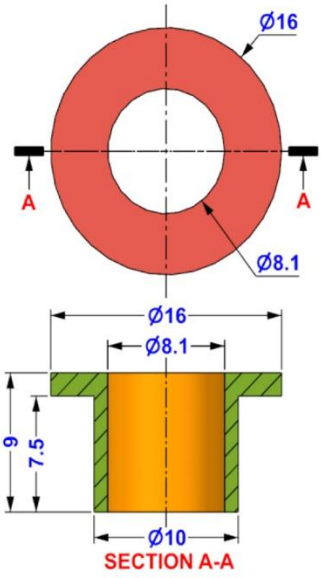
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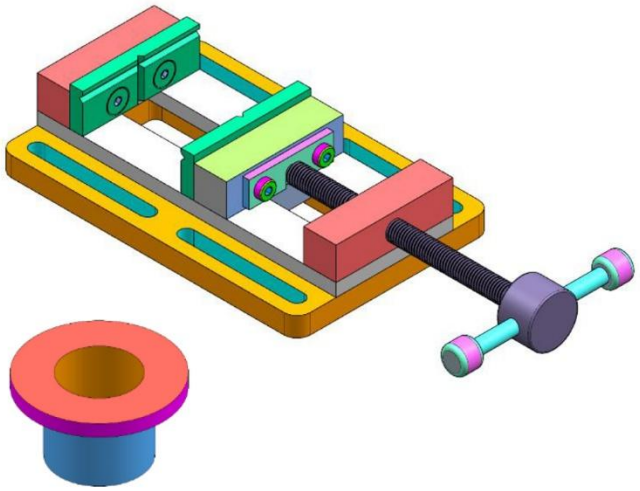
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- Assembly







SACHIDANAND JHA

SOLIDWORKS ALL IN ONE WORKBOOK

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• 2D Sketching • 3D Modeling • Assembly Drawings

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This book contains over 500 carefully crafted practice drawings, including:

- 200 2D Sketching Exercises
- 200 3D Modeling Exercises
- Comprehensive Assembly Models with 150+ Individual Part Drawings

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Founder & CEO, CADIN360

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SOLIDWORKS ALL IN ONE WORKBOOK

- This book contains over 500 CAD practice exercises, organized as:

  1. 200 2D Sketching Exercises
  2. 200 3D Modeling Exercises
  3. Assembly Projects with 150+ Part Drawings

- This book is a practice workbook. It does not include step-by-step tutorials for creating 2D drawing, 3D models and Assembly.

- SI units (millimeters) are used for all dimensions.

- Third Angle Projection is used throughout this book.

- This book is for **SOLDIWORKS** and also suitable for Other Feature-Based Modeling Software such as Inventor, Catia, Fusion 360, NX, Solid Edge, AutoCAD, PTC Creo etc.

- Designed for students, engineers, drafters, and designers looking for extensive CAD practice using SolidWorks.

- The exercises cover a wide range of real-world modeling challenges—from simple sketches to complex assemblies—offering clear, concise, and structured drawing practice.

- Exercises are organized to gradually develop beginner to advanced-level design skills.

- Each exercise is self-contained, and can be completed independently.

- Assembly drawings follow industry standards to help improve visualization and multi-part modeling skills.

- All dimensions are in mm. Assume missing dimensions logically.

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HOW TO USE THIS BOOK

This book contains over 500 CAD practice exercises, designed for self-paced learning using SOLIDWORKS or any feature-based modeling software.

- 2D Sketching Exercises: Start here if you're a beginner or learning how to use the sketch environment.

- 3D Modeling Exercises: Follow after mastering sketching. Practice creating solid models using the provided dimensions.

- Assembly Drawings: Use after completing part models to understand multi-part assemblies, relationships, and constraints.

Tips for Best Use:

- Complete the exercises in order, or jump to any skill level you prefer.
- All dimensions are in millimeters.
- Where dimensions are missing, apply logic or practice estimation.
- This book is ideal for both students and professionals preparing for industry design work.

Note:

This book is available in multiple formats – **Black & White**, **Standard Color**, and **Premium Color** editions.

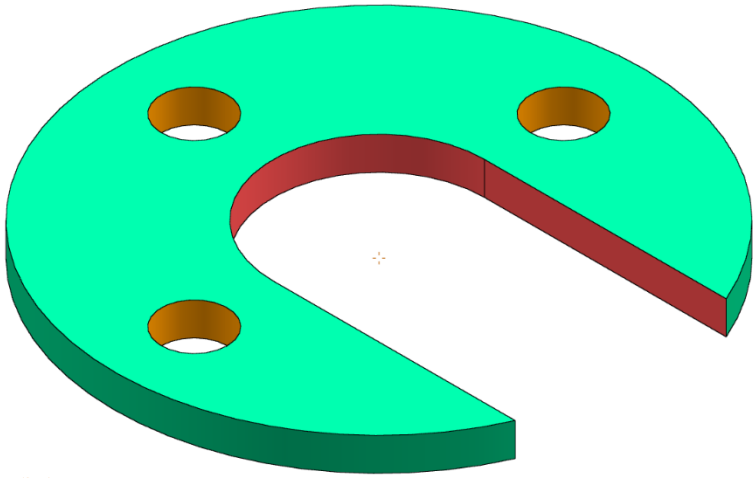
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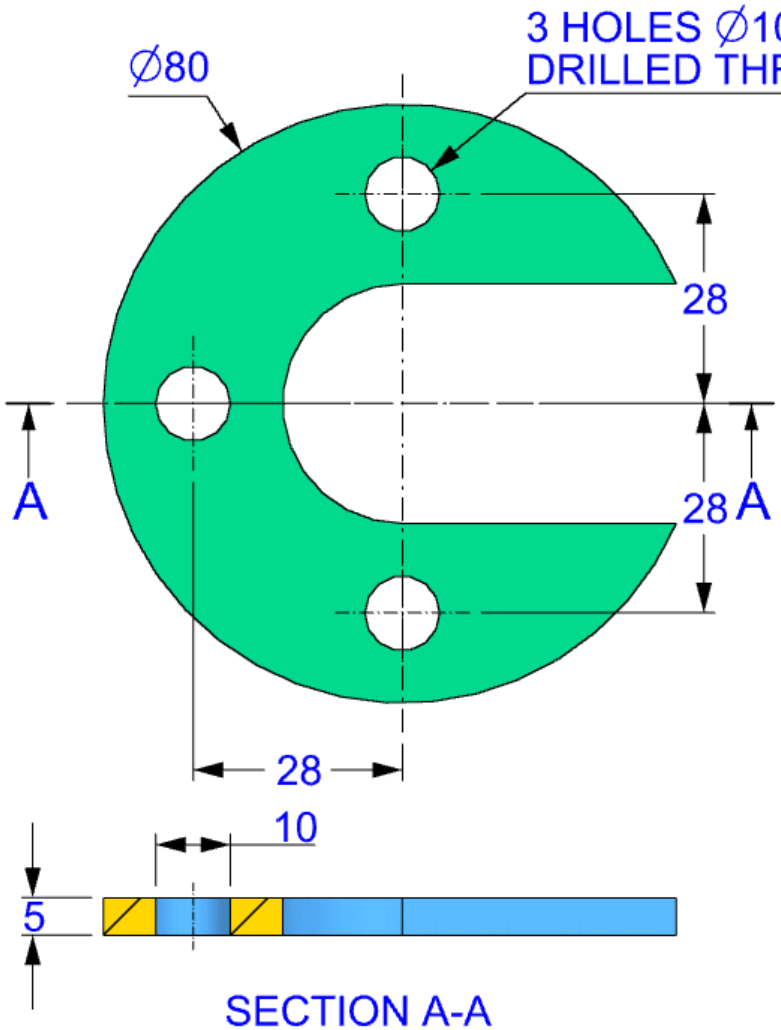
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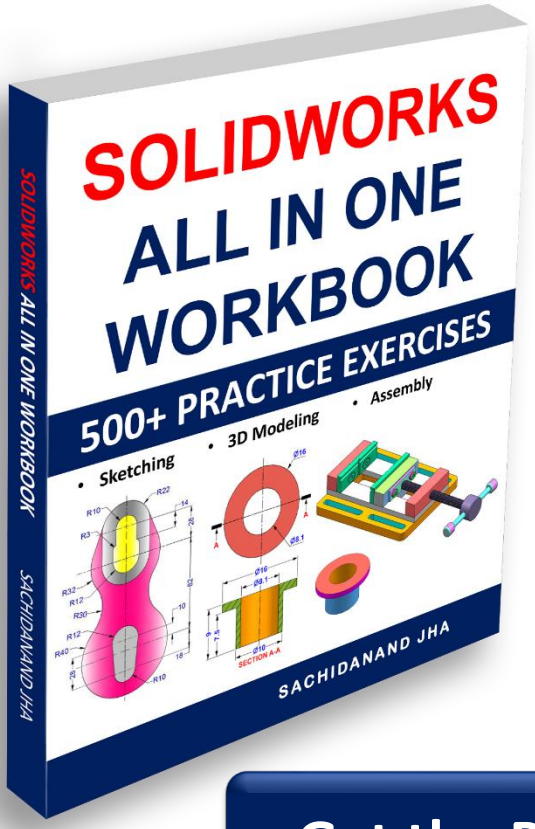
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- ✓ Books contains exercises of Sketching, 3D Modeling & Assembly.
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- ✓ Full Assembly STEP Files (.stp format) – Compatible with all major CAD software
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We're honored to be a part of your journey.
Happy Designing!
– Team **Cadin360**

Master SolidWorks with Real-World Practice Exercises

This book contains over 500 SolidWorks practice exercises including sketching, 3D modeling, and assembly drawings. Designed for students, engineers, and professionals to build practical CAD modeling skills.

SOLIDWORKS ALL IN ONE WORKBOOK

This book contains:-

- 200 2D Sketching Exercises
- 200 3D Modeling Exercises
- Multi-part Assembly Exercises & Detailed Drawings
- All drawings in 3rd Angle projection
- All dimensions are in mm(metric system)