

AUTODESK FUSION 360

2026

BLOG

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

HOW TO SUPPRESS FEATURES IN FUSION 360

• LEARN • • APPLY • • GROW •

Introduction

In Fusion 360, a powerful CAD/CAM software used by designers, engineers, and hobbyists alike, the ability to manage feature visibility and suppression is essential for efficient modeling.

Suppressing features allows you to temporarily hide or disable specific parts of your design, making complex models easier to work with, troubleshoot, or modify. Whether you're trying to speed up your workflow, analyze the impact of certain features, or prepare models for manufacturing, knowing how to suppress features in Fusion 360 is a fundamental skill. This guide provides a comprehensive, step-by-step approach to suppress features effectively—perfect for beginners and advanced users seeking to optimize their modeling process.

Understanding Features and Suppression in Fusion 360

Before diving into how to suppress features, it's important to understand what features are in Fusion 360. Features include sketches, extrudes, fillets, chamfers, holes, and other operations that modify the base geometry. Suppression temporarily disables these features without deleting them, offering flexibility for iterative design and troubleshooting.

Suppressing features is particularly useful when:

- You want to test how your model looks without certain features
- You need to simplify the model for simulation or analysis
- You're troubleshooting interference or fit issues
- You want to compare different design iterations quickly

Let's explore how to effectively suppress features in Fusion 360.

How to Suppress Features in Fusion 360: Step-by-Step Guide

Suppression is straightforward in Fusion 360 but can be confusing for new users. Follow these steps to master feature suppression.

1. Access the Browser Panel

The first step is to locate the feature you want to suppress in the Fusion 360 browser panel:

- Ensure the browser panel is visible. If not, click on "Browser" in the upper left corner.
- Expand the design tree to see all features, components, and bodies.

2. Select the Feature to Suppress

Identify the feature you wish to suppress:

- Click directly on the feature name (e.g., "Extrude," "Fillet," "Hole").
- For features within a multistep process, ensure you select the specific feature node.

3. Right-Click and Choose Suppress

Once selected:

- Right-click on the feature name.
- From the context menu, click on "Suppress" (sometimes labeled as "Suppress Feature").

4. Confirm Suppression

The feature will now appear grayed out or with a suppression icon, indicating it's inactive:

- The model immediately updates to reflect the suppression.
- You can suppress multiple features in succession for multiple comparisons.

5. Unsuppress a Feature

To re-enable a suppressed feature:

- Right-click again on the feature.
- Select “Unsuppress” from the context menu.
- The feature will regenerate, restoring the original model state.

6. Suppress Multiple Features at Once

For efficiency:

- Select multiple features by holding "Ctrl" (or "Cmd" on Mac) while clicking.
- Right-click any of the selected features.
- Choose “Suppress” to disable them all simultaneously.

Practical Example: Suppressing a Fillet

Suppose you added a fillet but want to see how the model looks without it:

- Find the fillet feature in the browser.
- Right-click and select “Suppress.”
- Observe the model update instantly.
- Unsuppress by right-clicking again when needed.

Practical Tips for Suppressing Features Effectively

- Use suppression to perform "what-if" analyses, such as removing holes or fillets to see their impact.
- Combine suppression with component visibility toggling for better model control.
- Remember that suppression temporarily disables features—they do not delete your work.
- Always save versions or backups before suppressing significant features, especially in complex models.

Best Practices When Suppressing Features

- **Plan Your Workflow:** Suppress features in a logical order—start with the most recent additions.
- **Document Your Changes:** Keep track of suppressed features for future editing.
- **Use the Timeline for Troubleshooting:** The timeline at the bottom shows feature history; right-click to suppress features directly from there.
- **Avoid Suppressing Critical Features:** Suppressing essential features may cause downstream failures. Be cautious.

Common Mistakes and How to Avoid Them

Mistake	How to Avoid It
Accidental suppression of critical features	Double-check the feature before suppressing
Forgetting to unsuppress features later	Make a habit of reviewing suppressed features before finalizing models

Suppressing features that impact complex assemblies

Suppress features step-by-step, testing assembly fit after each suppression

Advanced Techniques: Using the Timeline for Suppression

Fusion 360's timeline offers a visual sequence of features:

- Right-click on a feature within the timeline
- Select "Suppress" directly from there
- To unsuppress, right-click and select "Unsuppress"

This method provides precise control over feature suppression and is especially useful in complex models with multiple features.

Suppressing Features in Complex Assemblies

In assemblies:

- Suppressing individual parts often improves performance.
- For features within components, open the component context.
- Use the browser to suppress features or entire components as needed.
- This method aids in troubleshooting interference issues or optimizing assemblies.

Comparison: Suppressing vs. Deleting Features

Aspect	Suppressing	Deleting
Reversibility	Temporary, can be undone	Permanent, must be recreated
Usage	Templating, testing modifications	Final cleanup or removal
Caution	Safer for iterative editing	Risk of losing critical data

Suppression is generally preferred during iterative design, whereas deletion is suitable when features are no longer needed.

Conclusion

Suppressing features in Fusion 360 is a vital technique for efficient modeling, troubleshooting, and iterative design. By mastering this process, you gain greater control over your models, allowing for easier modifications and analysis. Remember to use suppression thoughtfully, leveraging the timeline and browser panel for best results. Practical understanding and proper application of feature suppression can significantly improve your Fusion 360 workflow, saving time and minimizing errors.

FAQ

1. How do I suppress a feature in Fusion 360?

Ans: Right-click on the feature in the browser panel and select “Suppress” from the context menu.

2. Can I unsuppress a feature after suppressing it?

Ans: Yes, right-click on the suppressed feature and choose “Unsuppress” to restore it.

3. Is suppressing features the same as deleting them?

Ans: No, suppression temporarily disables features without deleting them, allowing easy reactivation; deletion permanently removes them.

4. How do I suppress multiple features at once in Fusion 360?

Ans: Ctrl + click (Cmd + click on Mac) to select multiple features, then right-click and choose “Suppress” to disable all selected features simultaneously.

5. Can suppression affect downstream features?

Ans: Yes, suppressing earlier features can impact dependent features, so proceed with caution and check the model after suppression.

6. How do I manage suppressed features in complex models?

Ans: Use the timeline for precise control, right-click features for suppression, and monitor the model's behavior after each change for best results.

7. What are best practices for suppressing features?

Ans: Plan your suppression sequence, document changes, test frequently, and avoid suppressing critical features that could break downstream operations.

About CADIN360

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 Autodesk Fusion 360.

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

2026

Practice What You've Learned

You've just completed this blog and learned important concepts in Autodesk Fusion 360.

To help you practice and apply what you've learned, the next pages include a sample from our Fusion 360 book .This sample contains practice exercises and real-world practice tasks designed to strengthen your skills.

What you'll find next:

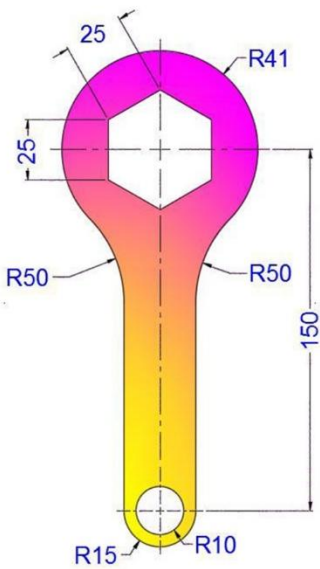
- ✓ Practice exercises from the book
- ✓ A brief overview of the complete book
- ✓ Options to explore or request the full sample

Your hands-on Fusion 360 practice starts next.

AUTODESK FUSION 360 ALL IN ONE WORKBOOK

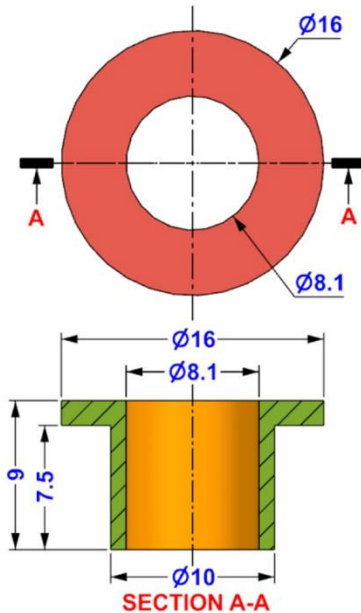
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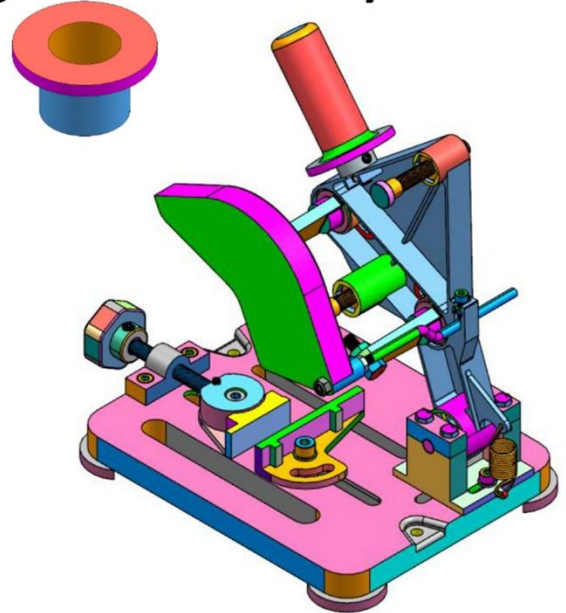
2D Sketching

• 3D Modeling



3D Modeling

• Assembly



Assembly

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500+ PRACTICE EXERCISES

2D Sketching • 3D Modeling • Assembly Drawings

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Dear Reader,

Thank you for choosing the AUTODESK FUSION 360 ALL IN ONE WORKBOOK. This book is part of the CADIN360° learning series, created to help engineers, students, and professionals master Fusion 360 through structured and practical exercises.

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

We founded CADIN360 in 2016 with the goal of delivering practical, high-quality learning material for CAD software. More than 9 years later, we're still committed to producing consistently exceptional books. With each of our titles, we're working hard to set a new standard for the industry. From the paper we print on, to the authors we work with, our goal is to bring you the best books available.

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Customer feedback is critical to our efforts at CADIN360.

Best regards,

Sachidanand Jha
Founder & CEO, CADIN360



AUTODESK FUSION 360 ALL IN ONE WORKBOOK

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AUTODESK FUSION 360 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 **AUTODESK FUSION 360** and also suitable for Other Feature-Based Modeling Software such as Inventor, Catia, SolidWorks, NX, Solid Edge, AutoCAD, PTC Creo etc.
- ❖ Designed for students, engineers, drafters, and designers looking for extensive CAD practice using Autodesk Fusion 360.
- ❖ 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.

HOW TO USE THIS BOOK

This book contains over 500 CAD practice exercises, designed for self-paced learning using Autodesk Fusion 360 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.

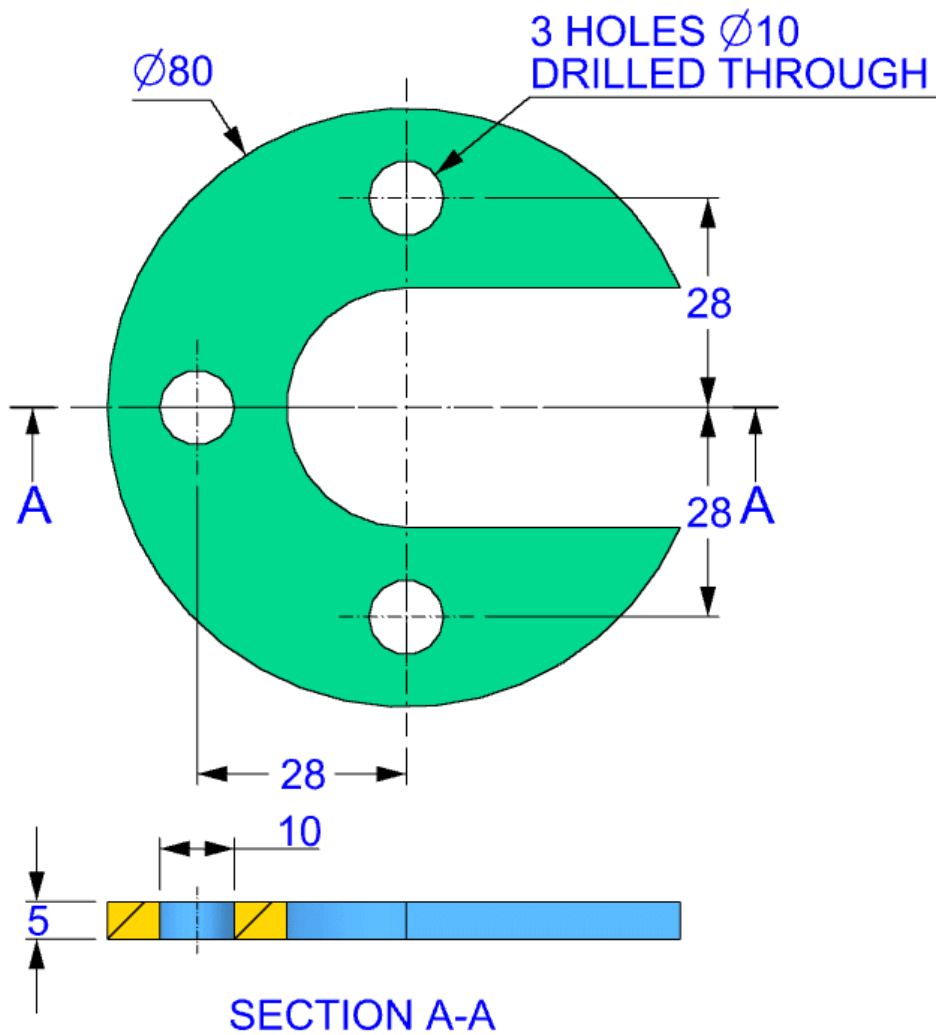
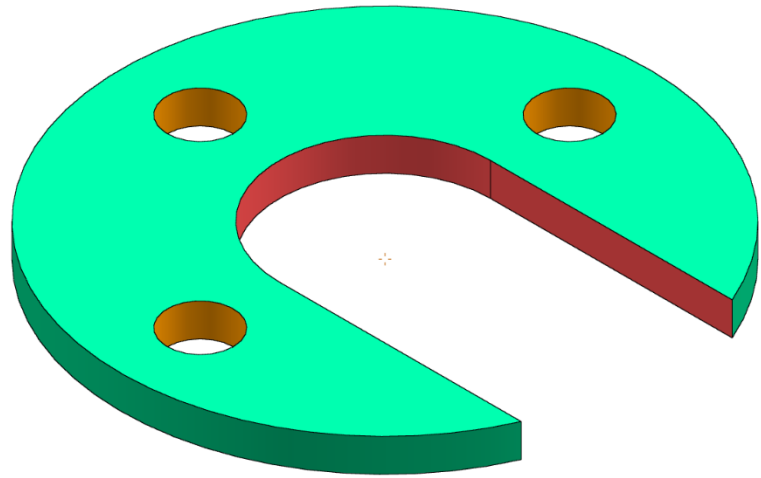
Note:

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

Happy learning!
– Team CADIN360

3D

EXERCISE-01



Get The Complete Practice Sample

You downloaded a single Exercise PDF

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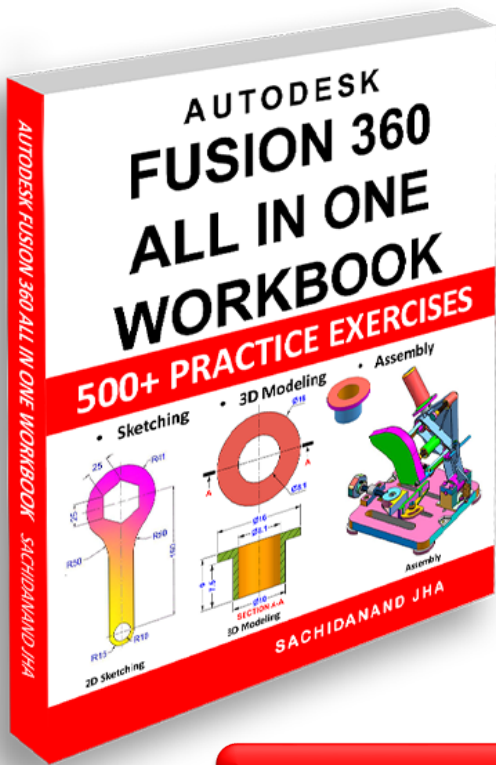
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What's Included in the **FUSION 360 ALL IN ONE WORKBOOK?**

- ✓ Books contains exercises of Sketching, 3D Modeling & Assembly.
- ✓ 500+ Practice Exercises with Dimensions
- ✓ Full Assembly STEP Files (.stp format) – Compatible with all major CAD software
- ✓ Get 200 3D Exercises in .f3d file format
- ✓ Get All Assembly Exercises in .STP file
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🚀 Keep Practicing. Keep Designing.

Whether you're a beginner or a pro, **practice is the key** to mastering any CAD software.

We're honored to be a part of your journey.

Happy Designing!

– Team **Cadin360**



Master Fusion 360 with Real-World Practice Exercises

This book contains over 500 Fusion 360 practice exercises including sketching, 3D modeling, and assembly drawings.

Designed for students, engineers, and professionals to build practical CAD modeling skills.

AUTODESK FUSION 360 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)