AUTODESK FUSION 360

2026

BLOG

A Note to Our Readers

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 EDIT REVOLVE LATER IN FUSION 360

• LEARN •   • APPLY •   • GROW •

Introduction

Revolving features are a fundamental component of parametric modeling in Fusion 360. They allow designers to create symmetrical or rotationally symmetric objects efficiently, reducing modeling time and ensuring precision. Sometimes, after initial creation, you may want to modify or refine the revolve without starting from scratch. Learning **how to edit revolve later in Fusion 360** is essential for flexible parametric design, especially when working on complex assemblies or iterative projects. This guide will walk you through the entire process with clear, step-by-step instructions, practical examples, and tips to ensure your workflow remains smooth and efficient.

Understanding Revolve in Fusion 360

Before diving into editing, it's important to understand what a revolve feature is in Fusion 360. Essentially, a revolve is a 3D feature created by rotating a 2D sketch or profile around an axis.

Types of Revolves:

- **Full revolve:** Creates a 360-degree symmetrical object.
- **Partial revolve:** Creates a revolved shape around a specified angle less than 360 degrees.

Why Edit a Revolve?

- Adjust the profile shape
- Change the revolve angle
- Modify the axis of rotation
- Fine-tune the feature for new design requirements

Knowing how to efficiently edit these features can save you time and streamline your design process.

How to Edit Revolve Later in Fusion 360: Step-by-Step Guide

1. Locate the Revolve Feature in the Timeline

Fusion 360 keeps a chronological history of your modeling steps in the timeline at the bottom of the interface.

- Find the revolve operation on the timeline.
- It's typically labeled as "Revolve" and appears as a specific icon.

2. Right-Click the Revolve in the Timeline

- Choose **"Edit Feature"** from the context menu.
- Alternatively, double-click directly on the revolve feature to open its dialog box.

3. Edit the Sketch Profile or Parameters

Once the feature editor opens, you have several options:

- **Modify the sketch profile:**
- Click **"Edit Profile"** to extract the sketch used.
- The sketch will open in the canvas, allowing you to modify the shape.

- **Change the revolve angle:**
- Adjust the angle in the revolve dialog box.
- For a full object, set it to 360°.
- For partial revolutions, input the desired angle.

- **Alter the revolve axis:**
- Select "Axis" in the dialog box.
- Choose a different edge, line, or create a new axis for rotation.

4. Rebuild the Sketch

- Make your edits.

- Finish the sketch by clicking **Finish Sketch**.

- Fusion 360 will automatically update the revolve based on your changes.

5. Use "Edit Feature" for Fine-Tuning

- If changes are minor, you can adjust parameters directly in the dialog box.

- For major modifications, editing the sketch ensures more control.

6. Confirm and Finish

- After making the adjustments, click **OK** in the dialog box.

- Fusion 360 regenerates the model with the applied changes.

Practical Examples of Editing Revolve Features

Example 1: Adjusting the Revolve Angle

Suppose you create a vase by revolving a profile 360°. Later, you decide to make it only half the height.

- Access the revolve feature.

- Change the angle from 360° to 180°.

- The model updates, reflecting the new partial revolution.

Example 2: Changing the Profile Sketch

You realize the profile outline has a mistake.

- Right-click the revolve in the timeline.

- Select **"Edit Feature"**.

- Click **"Edit Profile"** which opens the sketch.

- Correct the profile shape.

- Finish the sketch; the revolve updates accordingly.

Example 3: Modifying the Revolve Axis

Your original revolve axis placement isn't perfect.

- Edit the revolve feature.

- Select a different sketch line or edge as the new axis.

- Confirm; the model updates with the new axis of rotation.

Common Mistakes When Editing Revolve Later

- **Not selecting the correct feature in the timeline:** Ensure you right-click or double-click on the actual revolve feature.

- **Modifying the wrong sketch:** Double-check that you are editing the sketch used for the revolve.

- **Ignoring dependencies:** Changes in sketches or axes can affect other features; be cautious of dependencies.

- **Not regenerating the model:** Always click OK or Finish Sketch to apply changes.

Pro Tips and Best Practices for Editing Revolves in Fusion 360

- **Name your features:** Clear naming conventions in the timeline help quickly locate the revolve for editing.

- **Use parameters:** Define parameters for revolve angles or axes to facilitate easier updates.

- **Create multiple sketches:** For complex modifications, creating separate sketches can keep your workflow organized.

- **Save frequently:** Use versions or save backups before significant edits.

- **Utilize constraints:** When editing profiles, externally constrained sketches ensure predictable adjustments.

Comparison: Edit Revolve vs. Rebuild from Scratch

Aspect	Edit Revolve	Rebuild from Scratch
Time Efficiency	Faster; modifies existing feature	Slower; creates new feature from scratch
Parametric Control	Maintains history; adjustable parameters	Loses history; requires re-modeling

Flexibility	High; easy to fine-tune	Low; less adaptable
Risk of Errors	Lower; updates existing geometry	Higher; chances of inconsistencies

Editing existing revolve features in Fusion 360 offers a flexible, non-destructive way to refine your models, making your design process more efficient.

Conclusion

Knowing how to **edit revolve later in Fusion 360** unlocks greater control over your designs and saves valuable time. Whether refining profiles, adjusting angles, or changing axes, accessing and modifying the revolve feature through the timeline provides an intuitive and powerful method. Remember to leverage the edit feature, carefully select your sketches and parameters, and practice with real-world examples to become proficient. With these skills, you can confidently iterate your models, improve your workflow, and produce high-quality, parametric designs.

FAQ

1. How do I access the revolve feature for editing in Fusion 360?

Ans : Right-click on the revolve feature in the timeline and select "Edit Feature" or double-click it to open its parameters.

2. Can I change the sketch used in an existing revolve?

Ans : No, the sketch itself cannot be directly replaced, but you can edit the sketch to modify the profile used for the revolve.

3. How do I change the revolve angle after creating the feature?

Ans : Open the revolve feature's dialog box via "Edit Feature" and adjust the angle parameter accordingly.

4. Is it possible to rotate the revolve around a different axis later?

Ans : Yes, you can edit the revolve feature and select a different line or edge as the new axis.

5. Will editing a revolve feature affect other related features?

Ans : Yes, changes to a revolve feature can propagate downstream, affecting subsequent features depending on the design.

6. What should I do if my revolve feature isn't updating after edits?

Ans : Ensure you have confirmed and finished all sketch edits and click "OK" to regenerate the model.

7. Can I revert changes made during the editing process?

Ans : Yes, you can undo or roll back changes using Fusion 360's history or undo commands if needed.

About CADIN360

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.

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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

500+ PRACTICE EXERCISES

- **Sketching**
- **3D Modeling**
- **Assembly**

2D Sketching
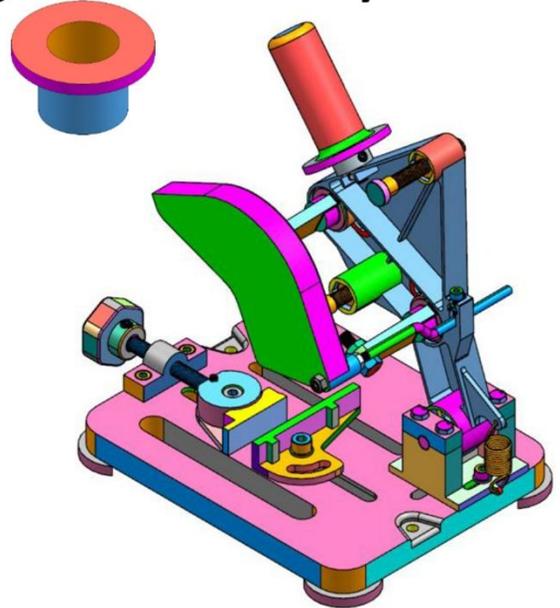
3D Modeling

Assembly

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2D Sketching • 3D Modeling • Assembly Drawings

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Learning Tutorials

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.

I hope you see all that reflected in these pages. I'd be very interested to hear your comments and get your feedback on how we're doing. Feel free to let me know what you think about this or any other CADIN360 book by sending me an email at cadin360@gmail.com

If you think you've found a technical error in this book, please visit https://cadin360.com/contact-us/.
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.
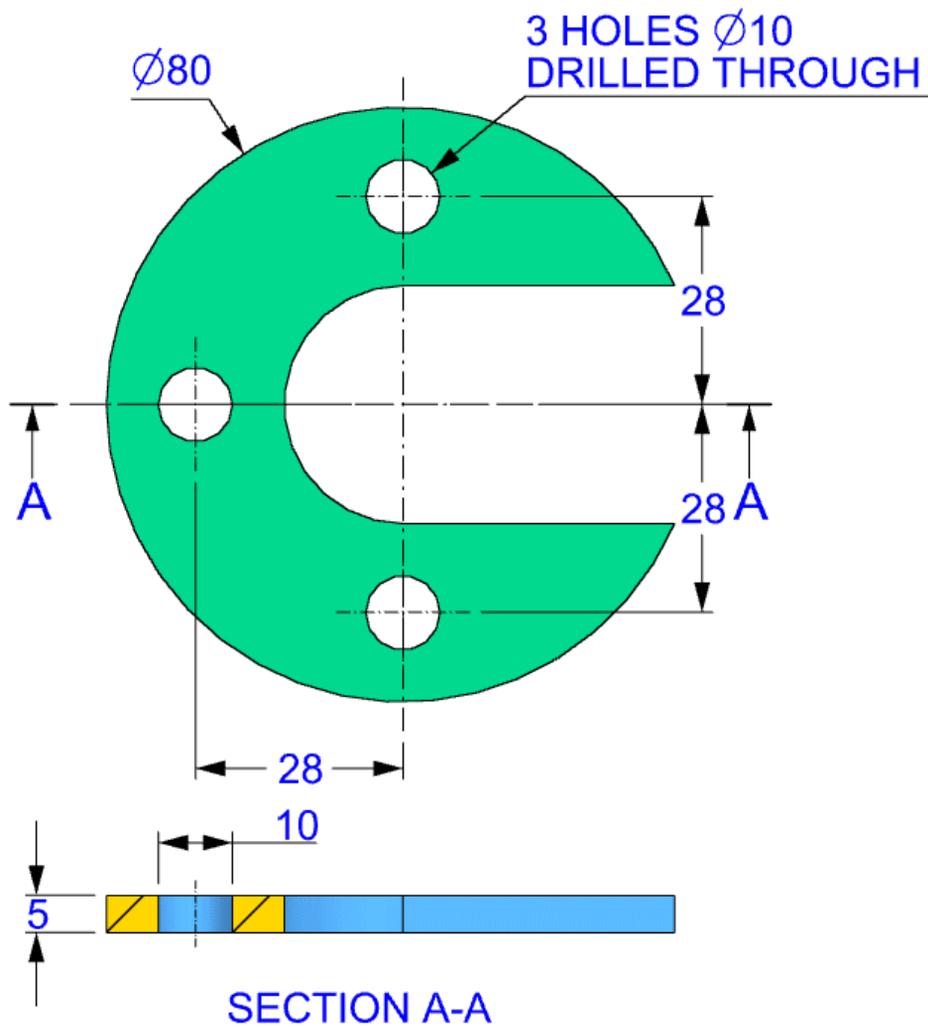- This book is ideal for both students and professionals preparing for industry design work.

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

Happy learning!
– Team CADIN360

3D

EXERCISE-01

Ø80

3 HOLES Ø10
DRILLED THROUGH

28

28 A

A

28

10

5

SECTION A-A

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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
- ✓ Instant Download Link - Sent to Your Email After Payment
- ✓ Lifetime Access to All Files

🧎 Thank You for Learning with Us!

Thank you for choosing the **AutoDesk Fusion 360 ALL IN ONE WORKBOOK**. We hope this book helped you strengthen your Fusion 360 skills through hands-on practice and real-world design challenges.

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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)