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 REVOLVE CUT MATERIAL IN FUSION 360

• LEARN •   • APPLY •   • GROW •

Introduction

Creating complex 3D objects often requires precise control over shapes, especially when working with symmetrical and rotational features. Fusion 360, a powerful CAD software, offers the "Revolve" tool that allows you to transform 2D sketches into intricate 3D models by revolving the sketch around an axis. Knowing how to properly revolve cut material in Fusion 360 is essential for efficient modeling, especially when designing mechanical parts, jewelry, or artistic objects. In this guide, we'll walk through the detailed steps to efficiently use the revolve cut feature, provide practical examples, and share tips to optimize your workflow.

Understanding the Basics of Revolve Cut in Fusion 360

Before diving into the step-by-step process, it's important to understand what a revolve cut is and how it differs from other features.

A revolve cut involves creating a sketch profile which is then rotated around an axis to remove material from the existing 3D model. Unlike extrusions, which extend the material outward, revolves cut material by spinning a profile around an axis—perfect for creating symmetrical holes or cutouts.

Fusion 360's revolve cut feature is highly versatile for:

- Creating circular or elliptical cutouts
- Designing rotational symmetrical parts
- Refining existing models with complex cut patterns

Step-by-Step Guide to Revolve Cut Material in Fusion 360

1. Prepare the 3D Model and Sketch

- Open Fusion 360 and load your existing model or start a new one.
- Select the face or plane where you want to create the cut.
- Create a 2D sketch on that face using the Sketch tools.

2. Draw the Profile for the Revolve Cut

- Use sketch entities like lines, arcs, or splines to draw the shape you want to cut.
- Remember, the profile should be a closed or open shape depending on the desired cut.
- **Tip:** To ensure a smooth and accurate cut, sketch your profile with clean, defined geometry.

3. Define the Axis of Revolution

- Draw or select the line or edge that will serve as the axis of revolution.
- This line should be straight and aligned properly to produce the desired cut.
- The axis can be part of the same sketch or an existing edge/face in your model.

4. Finish the Sketch

- Click "Finish Sketch" to exit sketch mode.
- Confirm that your profile and axis are correctly positioned.

5. Use the Revolve Cut Feature

- Switch to the "Solid" tab in the toolbar.
- Click on the "Create" dropdown and select "Revolve."
- In the "Revolve" dialog box, select the profile you just sketched.
- Choose the axis of revolution (either by clicking the axis line or selecting it from the browser).
- Set the angle for the revolve operation:

- **Full 360°** for complete rotation (creating a hole or complete cut).

- **Partial angle** for a segment or partial cut.

6. Set the Operation to Cut

- In the operation options, make sure to select "Cut."

- This ensures you are removing material rather than adding or creating new bodies.

- Confirm your settings and click "OK."

7. Inspect and Refine the Result

- Check the model in 3D view.

- Use the view cube or navigate to inspect the cut from different angles.

- Make adjustments if necessary by editing the sketch or changing revolve parameters.

Practical Examples of Using Revolve Cut in Fusion 360

Example 1: Drilled Hole in a Cylinder

Suppose you want to drill a through-hole in a cylindrical part:

- Sketch a circle on the face of the cylinder.

- Create a line through the circle to serve as the axis.

- Use the revolve cut to rotate the circle 360° around the axis, removing material in a clean, precise hole.

Example 2: Creating a Symmetrical Slot on a Gear

- Sketch the slot profile perpendicular to the gear's axis.

- Use the revolve cut to remove a segment, making the slot symmetrical and accurately placed.

Example 3: Increasing Complexity with Multiple Revolve Cuts

- Combine different profile sketches and axes.
- Use multiple revolve cuts to create intricate internal or external features, such as vents or decorative patterns.

Common Mistakes and How to Avoid Them

- Sketching the profile off-center: Ensure your profile is aligned with the intended axis.
- Forgetting to set the operation to "Cut": This can lead to creating added material instead of removing it.
- Using incomplete or disconnected sketches: Sketch clean, continuous profiles for seamless cuts.
- Over-rotating: Be cautious with angles—partial revolutions may be more appropriate for specific designs.

Best Practices for Using Revolve Cut in Fusion 360

- Always create the sketch on the correct plane or face aligned with your intended cut.
- Use construction lines for axes to keep sketches organized.
- Filter sketch entities to keep only what's necessary.
- Experiment with partial angles for design features like arcs or segments.
- Utilize the preview option before final confirmation — this helps see how the cut will look.

Comparison: Revolve Cut vs Other Removal Techniques

Feature	Description	When to Use	Pros	Cons
Revolve Cut	Rotates a profile around an axis to remove material	Symmetrical holes, segments	Precise, symmetrical cuts	Limited to rotational symmetry

Extruded Cut	Projects a sketch linearly through the model	Linear holes, profiles	Easy to use	Not suitable for circular cuts
Circular Pattern	Repeats a feature around a center point	Multiple holes or cutouts	Uniform pattern	Doesn't create a cut; used with other features

Conclusion

Mastering the revolve cut material in Fusion 360 empowers you to design complex, symmetrical objects efficiently. By following the step-by-step process—preparing the sketch, defining the axis, and customizing rotation angles—you can create precise cutouts and intricate features for a variety of projects. Whether designing mechanical parts, jewelry, or artistic components, the

revolve cut tool is an essential part of your CAD toolkit. Practicing these techniques and understanding common pitfalls will accelerate your proficiency and enhance your workflow.

FAQ

1. How do I create a perfect axis for revolved cuts in Fusion 360?

Ans: Use a construction line or existing edge aligned precisely with the intended rotational symmetry, ensuring it is a straight, clean, and well-positioned line.

2. Can I edit a revolve cut after creating it?

Ans: Yes, you can edit the sketch profile or parameter settings in the timeline or browser, and the revolve cut will update accordingly.

3. What is the maximum angle I can set for a revolve cut?

Ans: You can set any angle from 0° up to 360°, allowing for partial or full rotation as needed.

4. How do I make a partial revolved cut that only affects part of the model?

Ans: Use a partial angle less than 360° in the revolve dialog box to create segmental or arc-shaped cuts.

5. What are common mistakes when using the revolve cut feature?

Ans: Sketching profiles off the axis, forgetting to select "Cut" operation, or using incomplete profiles are common mistakes to avoid.

6. Can I combine multiple revolve cuts into one design?

Ans: Yes, you can create multiple sketches and perform separate revolve cut operations to compound complex features.

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.

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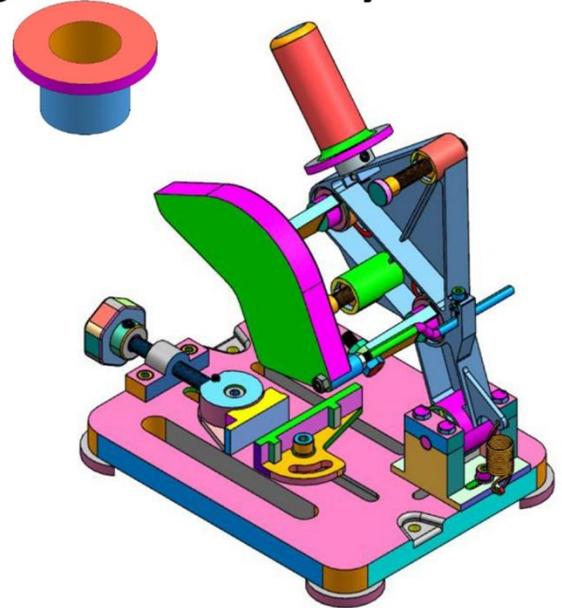
3D Modeling

Assembly

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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.

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

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Best regards,

Sachidanand Jha
Founder & CEO, CADIN360

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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.
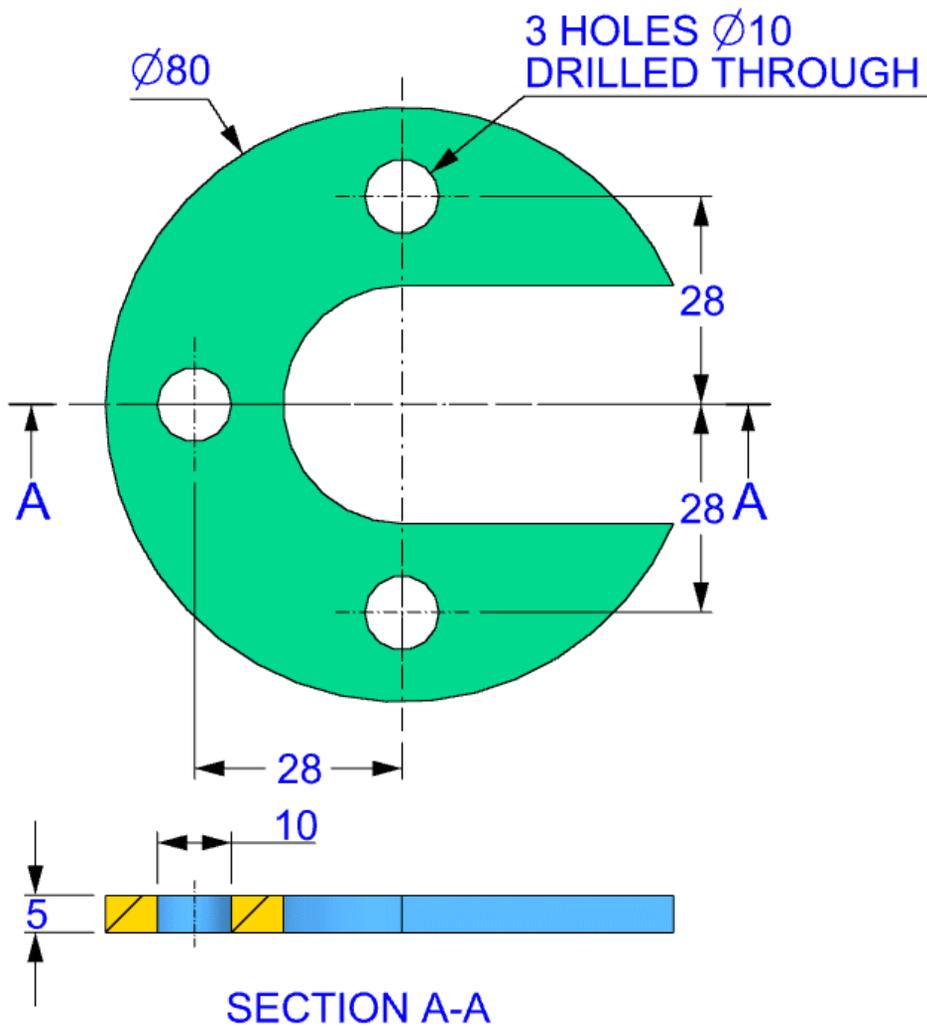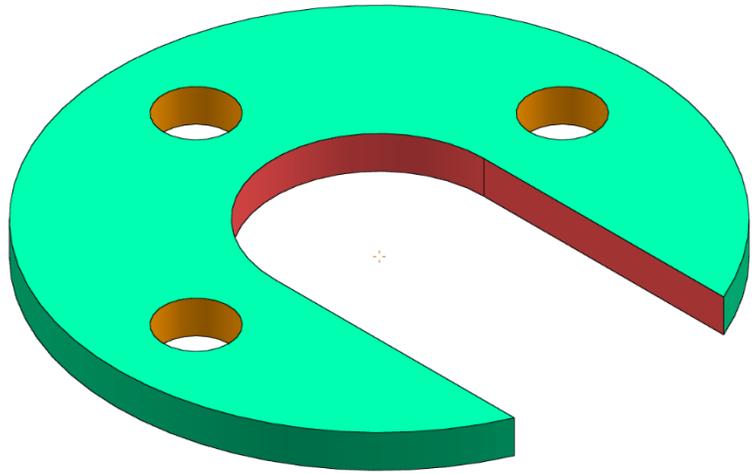- This book is ideal for both students and professionals preparing for industry design work.

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

Happy learning!
– Team CADIN360

3D

EXERCISE-01

3 HOLES ⌀10
DRILLED THROUGH

⌀80

28

28

A

28 A

28

10

5

SECTION A-A

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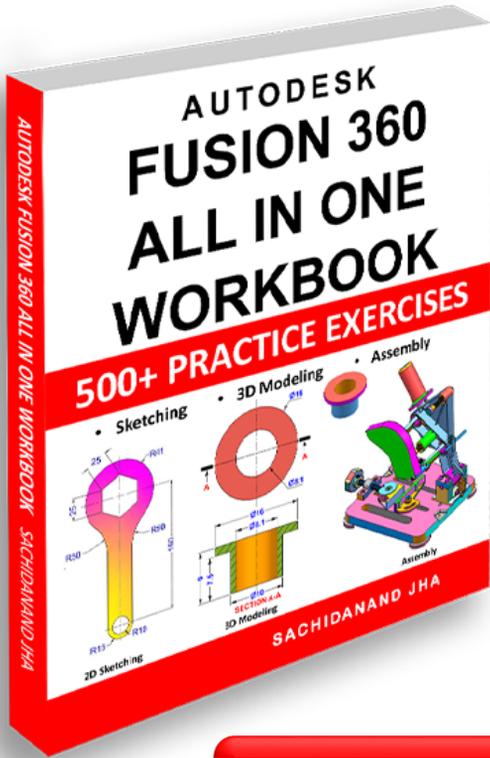
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🚀 Keep Practicing. Keep Designing.
Whether you're a beginner or a pro, **practice is the key** to mastering any CAD software. We're honored to be a part of your journey.
Happy Designing!
– Team **Cadin360**

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)