

AUTODESK FUSION 360

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

HOW TO REVOLVE HOLLOW SHAPE IN FUSION 360

• LEARN • • APPLY • • GROW •

Introduction

Creating hollow shapes in Fusion 360 is a fundamental skill that can significantly enhance your 3D modeling projects. Whether designing complex mechanical parts, artistic objects, or prototypes, the ability to revolve a hollow shape is vital for producing lightweight, sturdy, and visually appealing components. In this guide, we'll walk through how to revolve hollow shapes in Fusion 360 step-by-step, providing practical tips, common pitfalls to avoid, and real-world examples. By mastering this technique, you'll be able to streamline your workflow and improve the quality of your designs efficiently.

Understanding the Basics of Revolving in Fusion 360

Before diving into the creation process, it's essential to understand what "revolving" entails in Fusion 360. Revolve is a powerful feature that takes a 2D sketch and rotates it around an axis to create 3D objects like cylinders, cones, and more complex symmetrical shapes. When working with hollow objects, the key is to revolve a profile that has both outer and inner boundaries, creating a hollow shell.

Why Create Hollow Shapes?

- Lightweight structures in mechanical design.
- Artistic or decorative objects with internal cavities.
- Parts that require material reduction without sacrificing strength.
- Complex geometries that involve internal channels or spaces.

Revolving hollow shapes allows for precise control over wall thickness, internal cavities, and overall geometry, making it an indispensable technique.

Step-by-Step Guide: How to Revolve Hollow Shape in Fusion 360

Creating a hollow shape involves designing a profile that includes the inner and outer contours, then revolving it around an axis. Let's walk through this process.

1. Prepare the Sketch for the Hollow Profile

- Open Fusion 360 and start a new design or existing project.
- Create a new sketch on the XY plane (or any plane suited to your design).

2. Draw the Outer Profile

- Use the sketch tools (Line, Arc, Circle) to draw the outer shape.
- For a simple hollow cylinder:
 - Draw a circle representing the outer diameter.
- For complex shapes:
 - Sketch the profile that defines the outer boundary, considering the overall shape.

3. Draw the Inner (Hollow) Profile

- Inside the same sketch, draw the inner profile:
 - Use the Circle tool again, concentric or offset to the outer circle.
 - This inner circle represents the hollow cavity's inner surface.
 - Ensure the inner circle is smaller, defining the wall thickness.

4. Define the Axis of Revolution

- Identify the axis for revolution:
 - For a horizontal profile, draw a vertical or horizontal line as the axis.
 - Use the "Line" tool to sketch this axis inside or outside the profile.
 - For symmetry, the axis should pass through the center of the profile.

5. Finish Your Sketch

- Double-check all dimensions, especially the wall thickness (distance between inner and outer profiles).
- Fully constrain the sketch to avoid unintended movement.

6. Revolve the Profile

- Finish the sketch and go to the "Solid" tab.
- Select the "Revolve" tool.
- Click on the profile—make sure both outer and inner contours are selected.
- Choose the axis of revolution.
- Set the angle of revolution:
- Typically 360° for a full hollow object.
- Confirm the operation to generate the hollow shape.

7. Inspect and Refine Your Model

- Rotate the view to check the shape.
- Use the "Inspect" tool to verify dimensions, wall thickness, and internal cavity.
- Make adjustments to the sketch if necessary, then redo the revolve.

Practical Examples of Revolving Hollow Shapes

Example 1: Hollow Cylinder

- Draw concentric circles for outer and inner diameters.
- Revolve 360° around a central axis.
- Result: a hollow pipe with specified wall thickness.

Example 2: Hollow Vase with Curved Profile

- Sketch a profile with curves on one side.
- Use a vertical axis for revolution.
- Create a vase with a thin wall and intricate shape.

Example 3: Hollow Mechanical Part with Internal Channels

- Design complex internal cavities by sketching multiple profiles.
- Combine revolved shells with cut features for internal channels.

Common Mistakes to Avoid

- **Incorrect Axis Selection:** Ensure the axis truly passes through the center for symmetrical revolved shapes.
- **Unlinked Inner and Outer Profiles:** Sketch inner and outer boundaries separately before revolving to prevent errors.
- **Misaligned Profiles:** Fully constrain profiles to avoid off-center or skewed hollow shapes.
- **Incomplete Profile Sketch:** Make sure the sketch is closed, and all inner and outer profiles are connected.
- **Neglecting Wall Thickness:** Verify that the distance between inner and outer profiles meets design requirements.

Tips and Best Practices

- Always fully constrain your sketches to prevent accidental modifications.
- Use construction lines to define the axis clearly.

- For complex shapes, consider creating multiple sketches or using lofting techniques alongside revolved features.
- Use the "Inspect" tools regularly to verify dimensions.
- Save iterative versions to revert if needed.

Comparing Revolving Hollow vs. Solid Shapes

Feature	Revolving Hollow Shape	Revolving Solid Shape
Construction Method	Sketch inner and outer profiles, revolve	Sketch outer profile only, revolve solid
Material Usage	Less material, lightweight	More material, solid object

Internal cavities	Present, customizable	Absent
Design complexity	Slightly more involved due to inner profile	Simpler for basic shapes
Typical applications	Pipes, shells, artistic objects	Solid mechanical parts, solid models

Conclusion

Revolving hollow shapes in Fusion 360 is a versatile technique that empowers designers and engineers to create intricate, lightweight, and functional models. By carefully sketching concentric profiles, selecting the correct axis, and revolved a full 360°, you can produce complex hollow geometries suitable for various real-world applications. Practice, attention to detail, and understanding the underlying principles will help you master this essential skill, enabling you to elevate your 3D modeling projects confidently.

FAQ

1. How do I create a hollow cylinder in Fusion 360?

Ans : Sketch concentric circles for the outer and inner diameters, then revolve the profile 360° around the central axis to create a hollow cylinder.

2. Can I make a hollow shape without sketching the inner profile?

Ans : No, you need to sketch both inner and outer profiles or use a shell feature after creating a solid.

3. How do I ensure uniform wall thickness when revolved?

Ans : Draw concentric circles with the desired gap between them, ensuring the distance represents your wall thickness, and revolve the combined profile.

4. What is the best way to prevent the hollow shape from being off-center?

Ans : Fully constrain the profile and make sure the axis of revolution passes through the center of the sketch.

5. How do I hollow out an existing solid shape in Fusion 360?

Ans : Use the "Shell" command to hollow out a solid by specifying wall thickness or create sketches and subtract internal features with cut operations.

6. Is it possible to revolve only a partial segment for a hollow shape?

Ans : Yes, select the segment of the profile and set the revolve angle to less than 360° to create partial hollow shapes.

7. What common mistakes cause errors in hollow revolved parts?

Ans : Misaligning the axis, incomplete sketches, unlinked inner and outer profiles, and neglecting proper constraints are common causes.

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

500+ PRACTICE EXERCISES

• Sketching



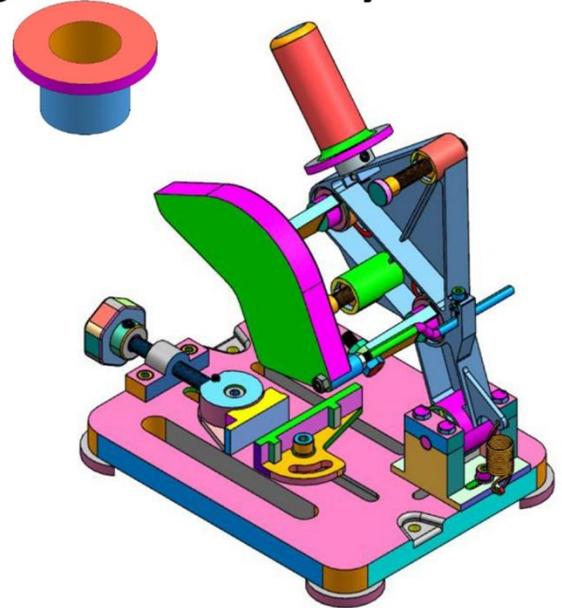
2D Sketching

• 3D Modeling



3D Modeling

• Assembly



Assembly

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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

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



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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.

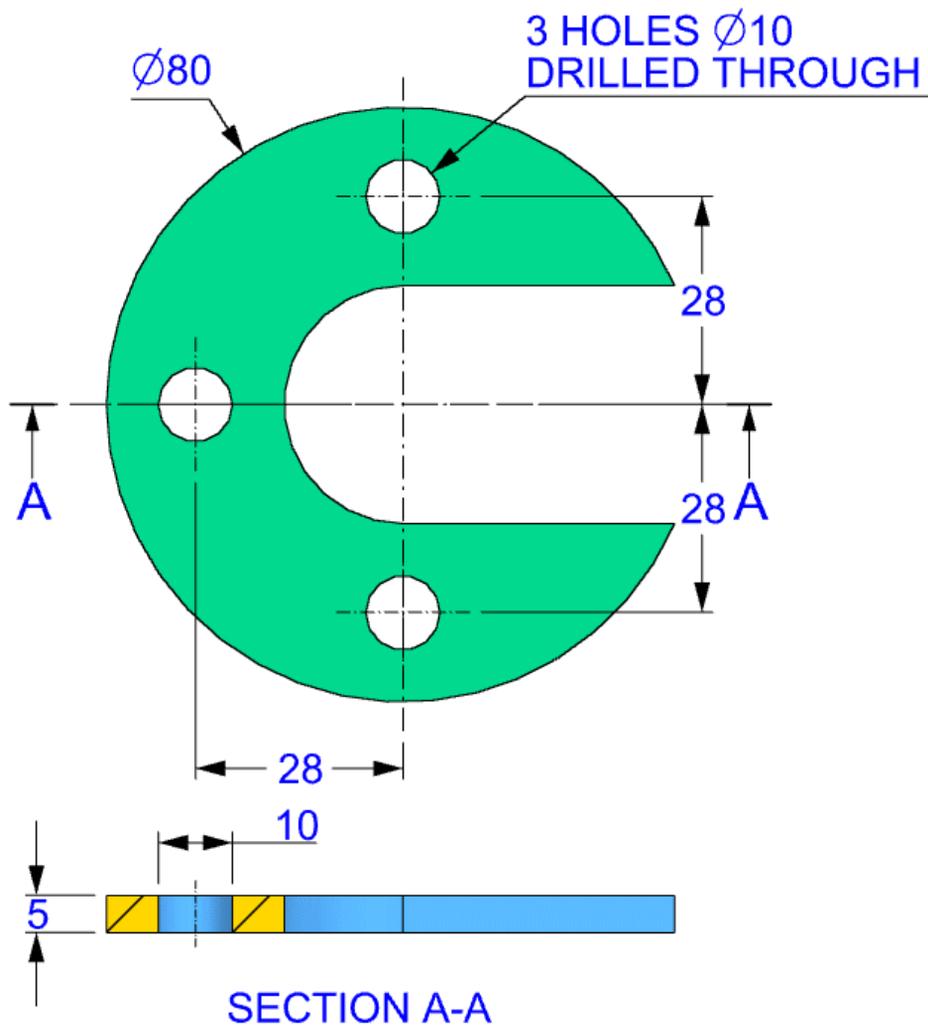
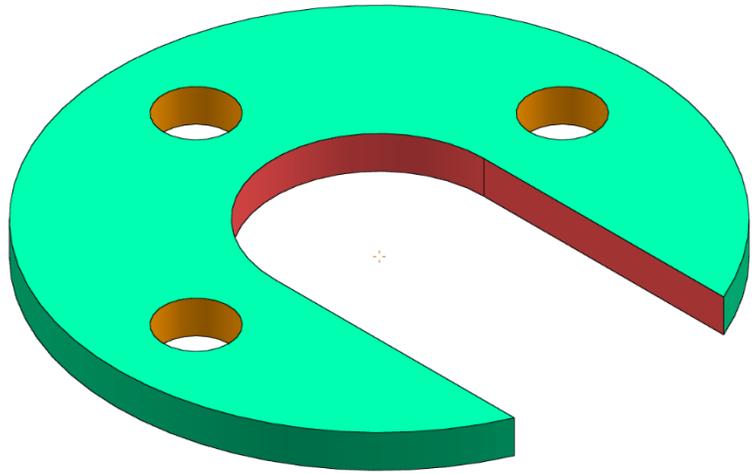
Note:

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

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– Team CADIN360

3D

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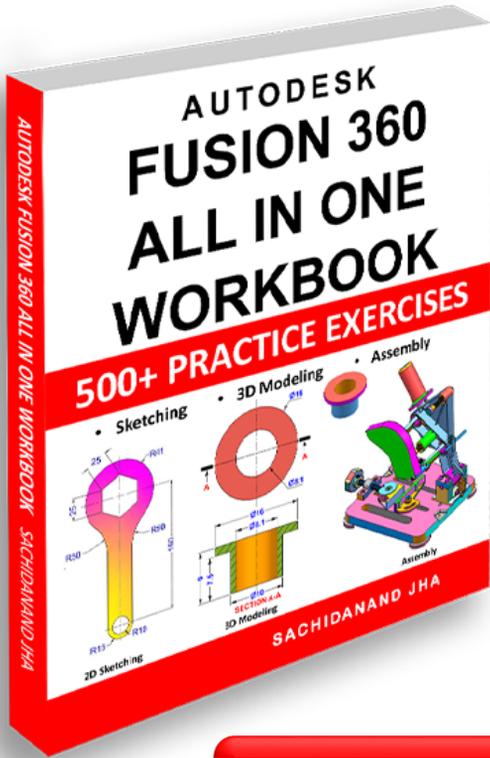
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🚀 Keep Practicing. Keep Designing.

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)