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 CREATE CYLINDER USING REVOLVE IN FUSION 360

• LEARN • • APPLY • • GROW •

Introduction

Creating a cylinder in Fusion 360 by revolving a sketch is a fundamental skill that opens up countless design possibilities. Whether you're designing mechanical parts, containers, or decorative objects, understanding how to use the revolve feature effectively is essential for efficient modeling. This guide will walk you through the entire process of creating a cylinder using revolve in Fusion 360, with step-by-step instructions, tips, and best practices for novices and experienced users alike.

How to Create a Cylinder Using Revolve in Fusion 360

In Fusion 360, the revolve feature allows you to create symmetric 3D objects by rotating a 2D sketch around an axis. This process can be particularly useful for crafting precise and complex cylinders. Let’s explore how to do this in a detailed, beginner-friendly manner.

Step 1: Set Up a New Canvas

Before diving into sketching, ensure your workspace is prepared:

- Launch Fusion 360 and open a new design.
- Save your project with a clear name for easy tracking.

Step 2: Create a Sketch on a Suitable Plane

The first step involves sketching the profile of the cylinder:

- Click on **Create Sketch** from the toolbar.
- Select the plane where you want to sketch (commonly the XY plane for vertical cylinders).

Step 3: Draw the Profile of the Cylinder

To revolve a shape into a cylinder, you need a 2D profile that, when rotated, forms the circular cross-section:

- Use the **Center Diameter Circle** tool:
- Click on **Center Diameter Circle**.
- Click on the origin point to set the circle's center at the origin.
- Drag outward to define the circle's radius.
- Enter the diameter of your desired cylinder.

Alternatively, you can draw a simple rectangle for a hollow or complex profile, but for a standard cylinder, a circle suffices.

Step 4: Define the Axis of Revolution

The axis of revolution is critical for generating the cylinder:

- Draw a straight line along the axis of the circle:
- Use the **Line** tool.
- Position it vertically through the center of the circle.
- Make sure the line extends beyond the circle's diameter to define the full length of the cylinder.
- Ensure the line is coincident with the center of the circle for symmetry.

Step 5: Finish the Sketch

Once your circle and axis line are ready:

- Click **Finish Sketch**.
- Verify your sketch looks correct, with the circle centered on the axis line.

Step 6: Use the Revolve Tool to Create the Cylinder

Now, transform your 2D profile into a 3D cylinder:

- Select **Create** from the toolbar, then choose **Revolve**.
- Click the profile (the circle) to select it.
- For the **Axis of Revolution**, select the line you drew.
- Set the **Angle** to 360° to create a full cylinder.
- Click **OK** to generate the object.

Your model is now a perfect cylinder created by revolving a circle.

Practical Examples of Creating Cylinders with Revolve

Creating cylinders via revolve is ideal for various real-world applications:

- **Mechanical Shafts**: Producing precise shafts with specific diameters and lengths.
- **Hollow Pipes**: Designing hollow cylinders by sketching two concentric circles and revolving the profile.
- **Container Bodies**: Creating cans or bottles with uniform cross-sections.
- **Decorative Elements**: Crafting columns or cylindrical ornaments with intricate profiles.

The versatility of the revolve method allows you to customize profiles for more complex shapes beyond simple cylinders.

Common Mistakes and How to Avoid Them

When creating a cylinder with revolve, here are common pitfalls and tips to prevent them:

1. **Incorrect Axis Placement**

- Mistake: Drawing the axis off-center or not aligned with the profile.

- Solution: Always ensure the axis is passing through the center of the circle profile and aligned correctly.

1. **Incomplete Profile**

- Mistake: Forgetting to fully define the profile or leaving it open.
- Solution: Use fully constrained sketches and closed profiles for revolved features.

1. **Wrong Revolution Angle**

- Mistake: Revolution angle less than 360°, creating partial or segmental shapes.
- Solution: Set the angle to 360° for a complete cylinder unless designing a segment or partial feature.

1. **Sketching on the Wrong Plane**

- Mistake: Drawing the profile on a non-relevant plane, leading to unexpected results.
- Solution: Choose the XY plane or appropriate reference plane aligned with your design intent.

Best Practices and Pro Tips

- **Use Constraints for Precision**
- Constrain your circle and lines to the origin or other reference points for accurate sizing.
- **Parametric Design**
- Use dimensions linked to parameters for easy adjustments later.
- **Exploit Symmetry**
- Drawing the profile and axis symmetrically reduces errors and simplifies modifications.

- **Start with a Simple Profile**
- For more complex shapes, build from simple profiles and modify as needed.

- **Test Revolve with Different Angles**
- Experiment with less than 360° for partial cylinders or segments to create unique features.

Comparing Revolve and Extrude for Creating Cylinders

While revolve is a powerful tool for creating symmetrical shapes from profiles, sometimes extrusion offers a more straightforward approach:

Method	Strengths	Best Use Cases
Revolve	Creates symmetrical, circular cross-sections from a profile	When designing objects around an axis, such as shafts or bowls

Extrude	Extends a 2D profile in a straight line	For rectangular shapes or simple blocks and outlines

Choosing between them depends on the design complexity and the shape's symmetry.

Conclusion

Mastering how to create a cylinder using revolve in Fusion 360 is crucial for effective 3D modeling, especially for designing mechanical parts and symmetrical objects. By following this detailed step-by-step process, you can produce precise, customizable cylinders effortlessly. Remember to focus on accurate sketching, correct axis placement, and setting the right revolution angle. With practice, this technique becomes an essential part of your design toolkit, enabling you to craft complex shapes with confidence.

FAQ

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

Ans: Draw two concentric circles in your sketch and revolve the area between them around the axis for a hollow cylinder.

2. Can I modify the size of the cylinder after creating it?

Ans: Yes, you can edit the sketch dimensions or parameter values and then update the revolve feature to resize the cylinder.

3. What's the difference between Revolve and Sweep in Fusion 360?

Ans: Revolve rotates a profile around a fixed axis to create symmetrical objects, while Sweep follows a path to create complex shapes along curves.

4. How do I create a segment of a cylinder, like a 90-degree quarter cylinder?

Ans: Set the revolve angle to less than 360° (e.g., 90°) during the revolve operation to create partial cylinders.

5. Is it possible to create a tapered cylinder using revolve?

Ans: Yes, by sketching a profile with varying radii along the height and revolving it, you can create tapered or conical cylinders.

This comprehensive guide should empower you to confidently create cylinders via revolve in Fusion 360, unlocking new design possibilities!

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.

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Practice What You've Learned

2026

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

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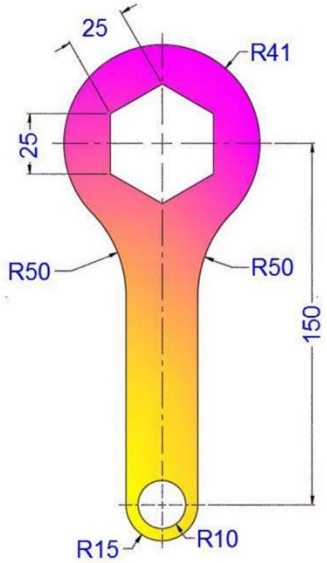
- Assembly



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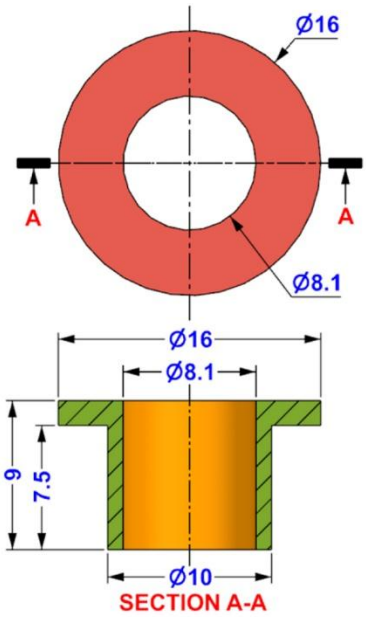

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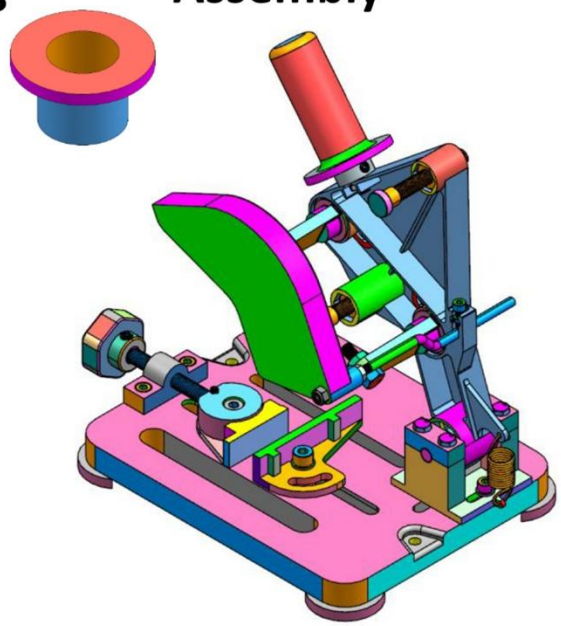

Assembly

SACHIDANAND JHA

AUTODESK

FUSION 360 ALL IN ONE WORKBOOK

500+ PRACTICE EXERCISES

2D Sketching • 3D Modeling • Assembly Drawings

SACHIDANAND JHA

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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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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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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.

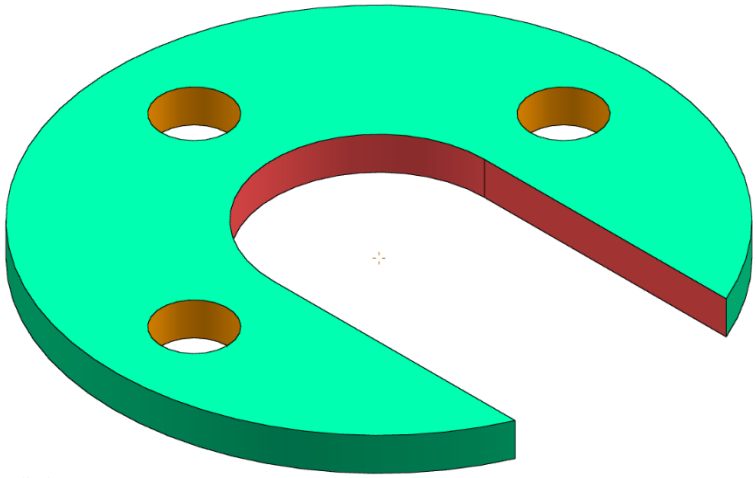
Tips for Best Use:

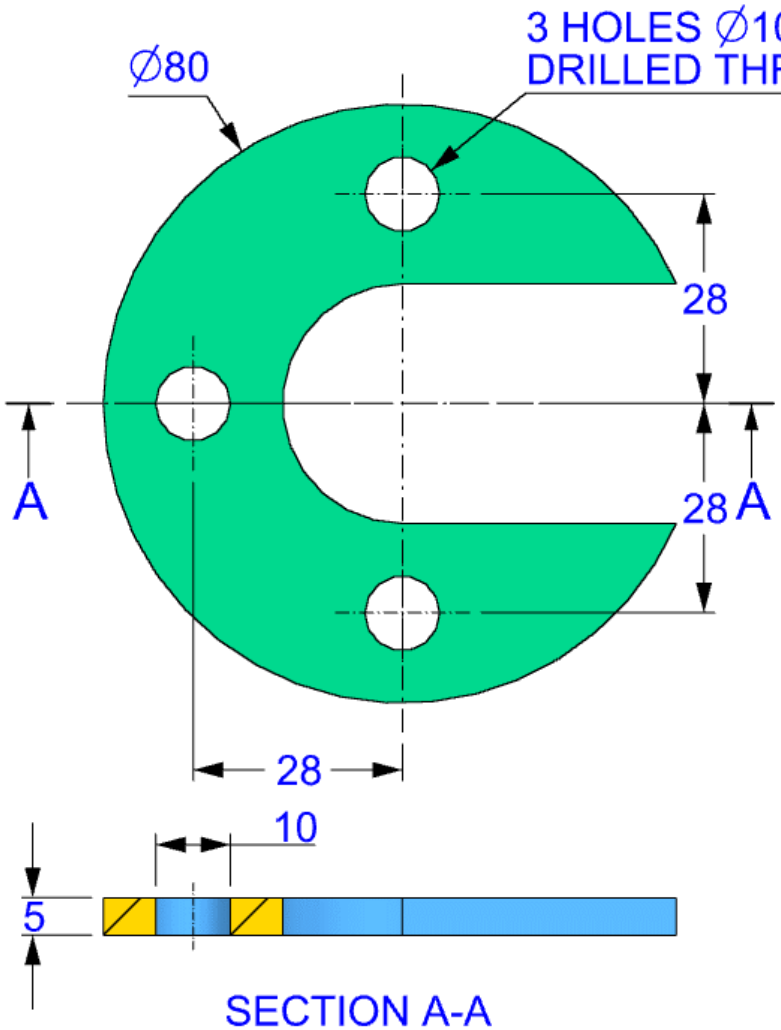
- Complete the exercises in order, or jump to any skill level you prefer.
- All dimensions are in millimeters.
- Where dimensions are missing, apply logic or practice estimation.
- This book is ideal for both students and professionals preparing for industry design work.

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

Happy learning!
– Team CADIN360

EXERCISE-01





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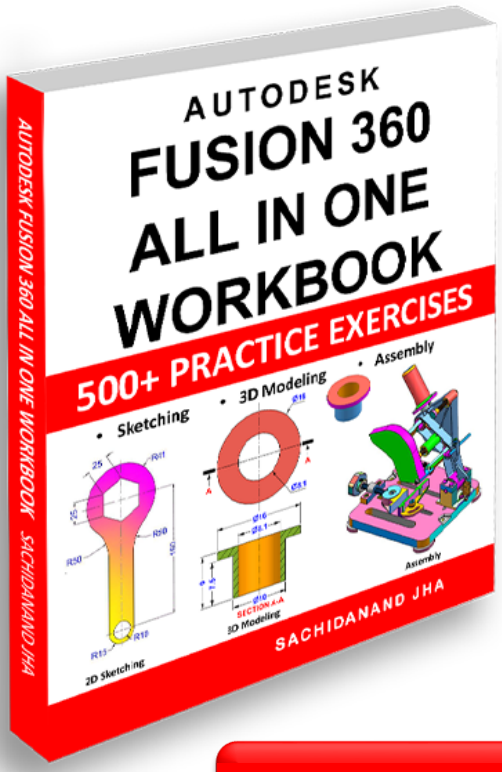

What's Included in the FUSION 360 ALL IN ONE WORKBOOK?

- ✓ 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
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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)