AUTODESK FUSION 360

2026

BLOG

www.cadin360.com

cadin360°
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.

Disclaimer:

— CADIN360 Team

WHEN TO USE REVOLVE INSTEAD OF EXTRUDE IN FUSION 360

• LEARN • • APPLY • • GROW •

Introduction

When designing 3D models in Autodesk Fusion 360, choosing the right tool for your task is essential for efficiency and precision. Two common methods to create solid geometry are revolve and extrude. While both serve to form 3D shapes, understanding when to use revolve instead of extrude can greatly improve your modeling workflow. This guide will explore the differences, applications, and best practices for leveraging the revolve feature, especially in contrast to extrude, to optimize your design process. Whether you're a beginner or looking to refine your skills, mastering the right use cases for each tool will help you produce cleaner, more accurate models.

Understanding the Fundamental Differences: Revolve vs. Extrude

Before diving into specifics, it’s vital to grasp what each feature does fundamentally:

- **Extrude:** Adds material along a straight path based on a 2D shape or profile, extending objects in one direction to create volume.
- **Revolve:** Creates a 3D shape by rotating a 2D profile around a defined axis, ideal for symmetrical objects with circular features.

Recognizing these distinctions forms the backbone for knowing why and when to use each tool.

When to Use Revolve Instead of Extrude in Fusion 360

The primary scenario to consider a revolve over an extrude is when your design involves objects with rotational symmetry or circular features. Here are detailed situations and practical examples:

1. Creating Symmetrical, Rotationally Symmetric Parts

Revolve excels in generating objects that are symmetrical about an axis.

- **Examples:**

- Vases, bottles, and cups
- Shafts and cylindrical housings
- Gear wheels and pulleys
- **Why choose revolve:**
- Instead of creating multiple sketches across different planes, you only need a single 2D profile.
- It ensures perfect symmetry, which is harder to achieve with multiple extrudes.

2. Modeling Hollow or Solid Rotational Components

Revolve can efficiently produce hollow shells and solid bodies.

- **Examples:**
- Hollowed out cylinders or pipes
- Complex turbine blades
- **Practical tip:** Create a profile that includes interior and exterior outlines, then revolve to form either a solid or hollow shape.

3. Designing Complex Curves with Symmetry

When working with complex, curved surfaces that revolve around an axis, using revolve can simplify the process.

- **Examples:**
- Beaded rings
- Ornamental objects with symmetric patterns
- **Comparison:** Using extrude for these shapes would require multiple cuts, chamfers, or lofts, making revolve simpler and cleaner.

4. Developing Revolved Mechanical Parts

Mechanical components with circular symmetry are best modeled with revolve.

- **Examples:**
- Cam profiles
- Threaded components
- Rotating shafts
- **Advantages:**
- Ensures precise symmetry
- Easy to modify by adjusting the sketch or axis

5. When the 2D Profile is a Half-Section or Segment of a Circle

Revolve is perfect for creating objects from a half-section of a circle or arc.

- **Examples:**
- Containers with rounded profiles
- Brake discs with curved surfaces
- **Note:** Instead of extruding a half-arc and then mirroring or trimming, revolving the arc simplifies the process.

How to Use Revolve in Fusion 360: Step-by-Step Guide

To effectively use the revolve feature, follow these practical steps:

1. Prepare Your Sketch

- Open Fusion 360.

- Create a new sketch on the plane that best aligns with your design.
- Draw the 2D profile, ensuring it spans from the rotational axis outward for symmetry.

2. Define the Axis of Revolution

- Select the line or edge that will serve as the axis.
- Make sure this line is clearly defined and runs through the center of your profile.

3. Apply the Revolve Tool

- Finish the sketch.
- Select the 'Create' menu, then choose 'Revolve'.
- Pick the profile you want to revolve.
- Select the axis line.
- Enter the angle of revolution: 360° for a full rotation, or less for partial features.

4. Adjust Parameters and Confirm

- Review the preview.
- Modify the angle if partial shapes are needed.
- Click 'OK' to generate the revolve feature.

5. Fine-Tune the Model

- Use fillets, chamfers, or shells to refine the shape.
- Combine with other features like cuts or holes for complex parts.

Practical Examples of Revolve in Action

Let’s look at real-world applications to reinforce understanding:

Part Type	Design Process	Revolve Advantage
Bottle	Sketch profile of half the side	Revolve 360° around the central axis
Gear	Draw tooth profile & revolve	Ensures precise symmetry and gear teeth placement
Shaft	Sketch the cross-section & revolve	Creates smooth, perfect rotational parts

These examples showcase the efficiency and accuracy revolved features bring to typical engineering components.

Common Mistakes to Avoid When Using Revolve

Even experienced designers can make errors. Here are typical pitfalls:

- **Incorrect Axis Selection:** Choosing a misguided axis can create distorted or unintended shapes.
- **Incomplete Profiles:** Omitting parts of the profile, especially near the axis, can lead to hollow or misshapen models.
- **Overcomplicating the Sketch:** Trying to include too many curves or details in a single profile can hinder the revolve operation.
- **Not Setting the Correct Angle:** Remember that the default is 360°. Adjust only when a partial shape is desired.
- **Ignoring Symmetry Constraints:** If the shape isn't symmetric, revolved features may not produce the expected geometry.

Best Practices for Using Revolve Effectively

To maximize your success with the revolve feature in Fusion 360:

- **Start Simple:** Keep your sketches clean and simple for ease of adjustment.
- **Use Construction Lines:** Draw reference axes to ensure precise rotation.
- **Exploit Symmetry:** Create profiles on one side and revolve to save time.
- **Validate the Axis:** Double-check the position and orientation of your axis before completing the revolve.
- **Combine with Other Features:** Use revolve in conjunction with cuts, holes, or shells for complex assemblies.

Comparing Extrude and Revolve: When to Choose Which

Feature	Best for	Limitations	Suitable Applications
Extrude	Creating straight, boxy, or complex non-symmetrical shapes	Less effective for rotational symmetry	Blobs, blocks, and objects with unique features

Revolve	Creating objects with circular, symmetric profiles	Less flexible for asymmetrical shapes	Cylinders, wheels, bottles, and mechanical parts

Understanding these distinctions guides you toward making smarter modeling choices.

Conclusion

Deciding when to use revolve instead of extrude in Fusion 360 hinges on understanding the geometry and symmetry of your design. Revolve is invaluable when creating parts with rotational symmetry, such as cylinders, gears, or curved objects, offering precision and efficiency that extrude cannot match. By mastering the proper application of revolve, along with best practices, you can streamline your workflow, produce cleaner models, and better meet design specifications. Remember, selecting the right tool—be it revolve or extrude—can significantly impact the quality and speed of your CAD projects.

FAQ

1. When should I use revolve instead of extrude in Fusion 360?

Ans: Use revolve when designing objects with rotational symmetry, such as cylinders, gears, or curved profiles, to ensure perfect symmetry and efficiency.

2. Can I create complex shapes using only revolve in Fusion 360?

Ans: While revolve is ideal for symmetric parts, complex non-symmetrical shapes may require a combination of revolve, extrude, loft, or other features.

3. How do I create a hollow part using revolve?

Ans: Draw a profile representing the outer and inner contours of the hollow section, then revolve it to produce the shell or hollow form.

4. What is the best way to ensure symmetry when using revolve?

Ans: Use a construction axis line passing through the center of your profile during the sketch phase, and make sure your profile is symmetric relative to this axis.

5. Can revolve be used for partial shapes or only complete 360° objects?

Ans: Revolve can create partial objects by specifying an angle less than 360°, such as 180° or 90°, for semi- or quarter-revolutions.

6. Are there limitations to what can be modeled with revolve in Fusion 360?

Ans: Yes, for non-symmetrical or highly intricate shapes, other features like loft, sweep, or freeform may be more appropriate.

7. How does the revolve feature affect model editing later?

Ans: Revolving creates parametric features, so adjusting the original sketch or axis will update the revolved shape dynamically.

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.

www.cadin360.com | support@cadin360.com

cadin360°
Learning Tutorials

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

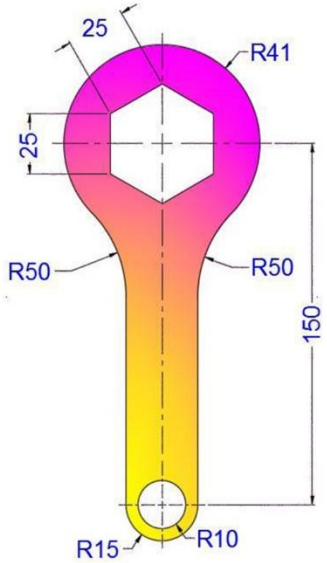
Your hands-on Fusion 360 practice starts next.

AUTODESK

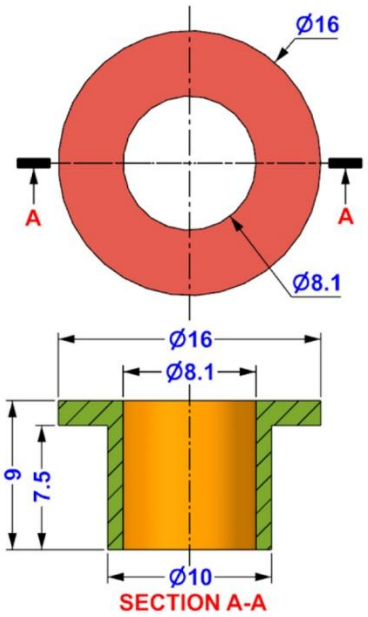
FUSION 360 ALL IN ONE WORKBOOK

500+ PRACTICE EXERCISES

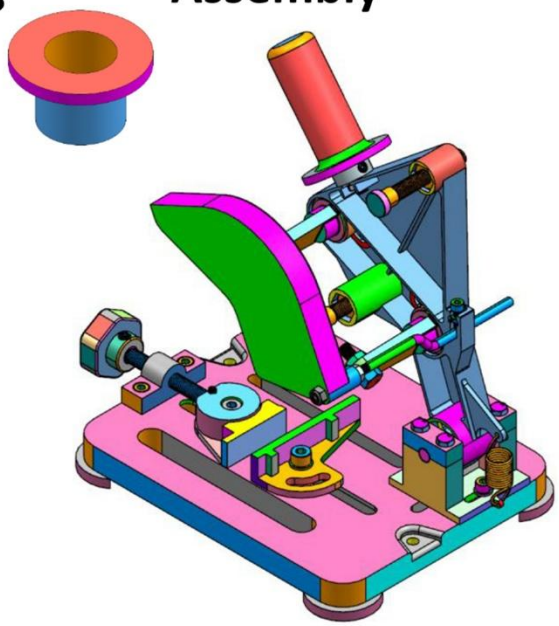
- Sketching
- 3D Modeling
- Assembly



2D Sketching



3D Modeling



Assembly

SACHIDANAND JHA

AUTODESK

FUSION 360 ALL IN ONE WORKBOOK

500+ PRACTICE EXERCISES

2D Sketching • 3D Modeling • Assembly Drawings

SACHIDANAND JHA

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

Published by CADIN360
Website: cadin360.com

Limit of Liability/Disclaimer of Warranty:

The publisher and the author make no representations or warranties with respect to the accuracy or completeness of the contents of this work and specifically disclaim all warranties, including without limitation warranties of fitness for a particular purpose. No warranty may be created or extended by sales or promotional materials. The advice and strategies contained herein may not be suitable for every situation. This work is sold with the understanding that the publisher is not engaged in rendering legal, accounting, or other professional services. If professional assistance is required, the services of a competent professional person should be sought. Neither the publisher nor the author shall be liable for damages arising herefrom. The fact that an organization or Web site is referred to in this work as a citation and/or a potential source of further information does not mean that the author or the publisher endorses the information the organization or Web site may provide or recommendations it may make. Further, readers should be aware that Internet Web sites listed in this work may have changed or disappeared between when this work was written and when it is read.

Examination Copies

Books received as examination copies in any form such as paperback and eBook are for review only and may not be made available for the use of the student. These files may not be transferred to any other party. Resale of examination copies is prohibited

Electronic Files & Usage Rights:

The electronic file/eBook in any form of this book is licensed to the original user only and may not be shared, distributed, resale or transferred to any other party. To access files, the user must contact **cadin360@gmail.com** with valid proof of purchase. Unauthorized distribution of the files is a violation of copyright law.

Disclaimer:

All product names, logos, brands, and registered trademarks mentioned in this publication are the property of their respective owners and are used for identification purposes only.

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:

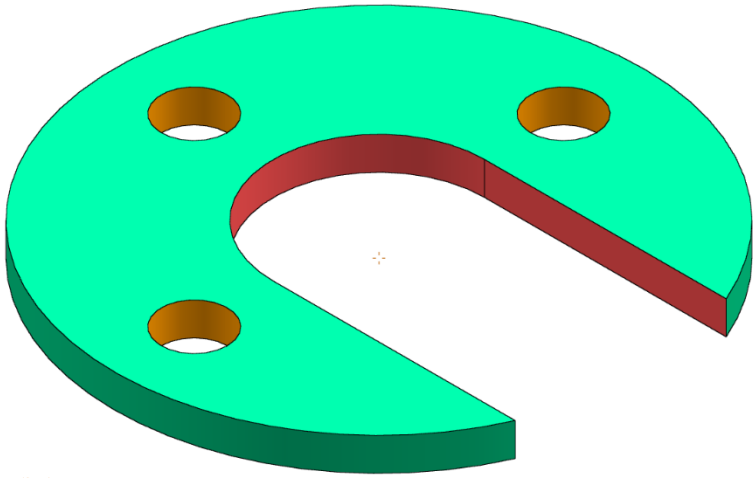
- Complete the exercises in order, or jump to any skill level you prefer.
- All dimensions are in millimeters.
- Where dimensions are missing, apply logic or practice estimation.
- This book is ideal for both students and professionals preparing for industry design work.

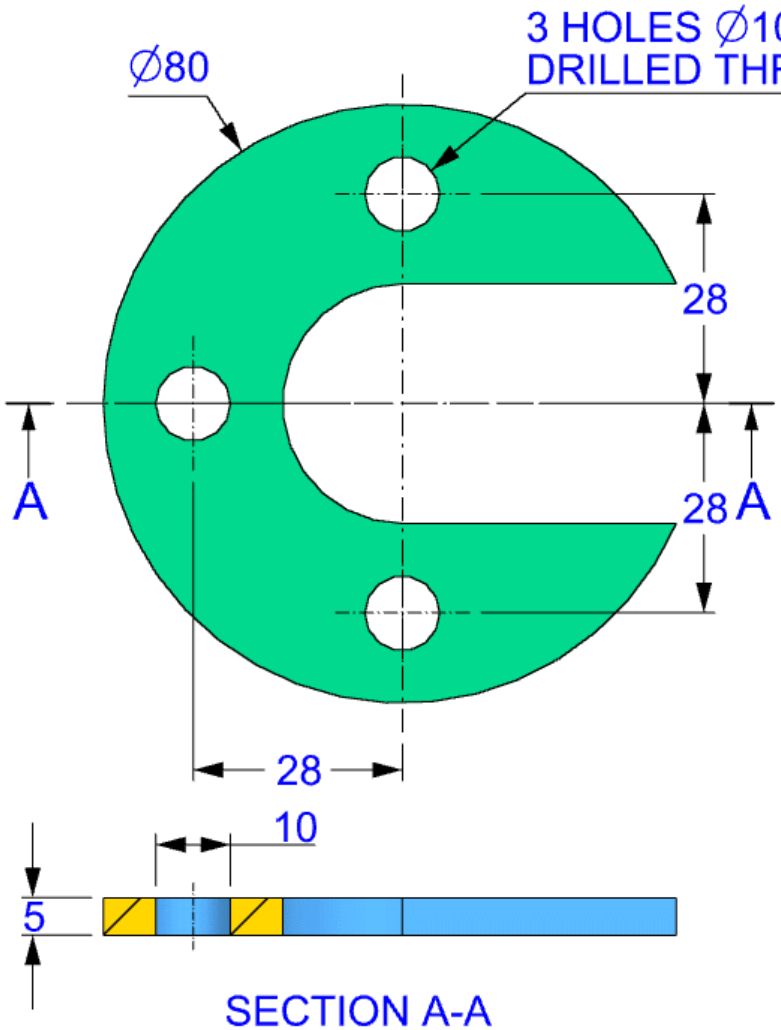
Note:

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

Happy learning!
– Team CADIN360

EXERCISE-01





Get The Complete Practice Sample

You downloaded a single Exercise PDF

The complete practice sample for this software includes multiple exercises and is not available inside this PDF..

What you will receive

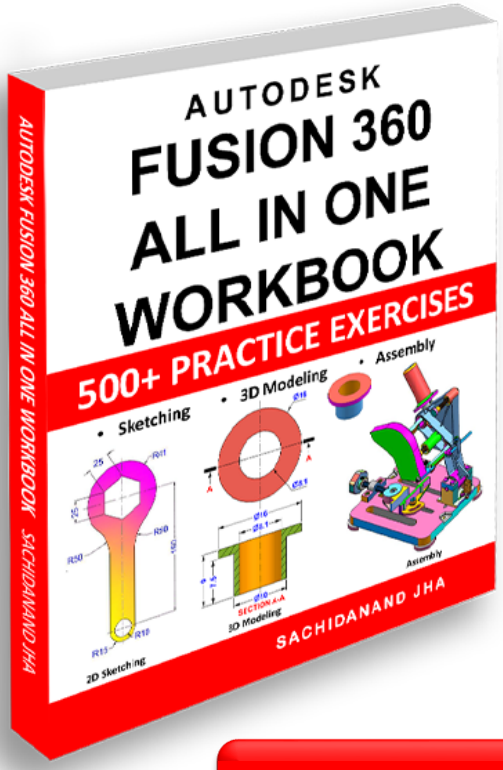
- A software-specific complete sample PDF
- Multiple real practice exercises (not a single file)
- Same quality as our professional training material
- Compatible with the latest software version

How to get the complete sample

Click the button below and **enter a valid email address**.
The **complete sample PDF will be delivered automatically** after the form is submitted.

SEND THE COMPLETE SAMPLE TO MY EMAIL

END OF SAMPLE



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

Get the Paperback book on Amazon

Get the Complete Bundle for Only $27.99

Special Offer for Students & Learners

Are you a Student, Unemployed or Financially struggling ?
Get this special Bundle only for $19.99

Special Offer for Only $19.99

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.
Your feedback means the world to us!
If you found this book helpful, please take a moment to leave a **review** on the Amazon where you purchased it. Your kind words not only motivate us but also help other learners discover our resources. Scan the QR.

⭐ A good review goes a long way!

📘 Explore More CAD Practice Books
Looking to continue your learning journey?
We offer similar practice-based books for over **30 CAD software platforms**, including:

- AutoCAD
- SolidWorks
- FreeCAD
- TinkerCAD
- TurboCAD
- Siemens NX
- CATIA
- Creo
- SketchUp and many more...

Visit our website 👉 **www.cadin360.com** to browse the complete collection.

💬 Stay Connected
Have suggestions, feedback, or just want to say hello?
We'd love to hear from you!
📧 Email: cadin360@gmail.com
🌐 Website: www.cadin360.com

🚀 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)