

AUTODESK FUSION 360

2026

# BLOG

 [www.cadin360.com](http://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:

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.

— CADIN360 Team

# WHAT REVOLVE COMMAND DOES IN FUSION 360

• LEARN •      • APPLY •      • GROW •

# Introduction

In the realm of CAD software, Fusion 360 has earned widespread acclaim for its powerful modeling tools and intuitive interface. Among its essential features is the revolve command, a fundamental tool used to create symmetrical, three-dimensional objects from two-dimensional sketches. Whether you're designing a bottle, a gear, or a custom mechanical part, understanding how the revolve command works in Fusion 360 is crucial for turning your ideas into precise digital models. In this comprehensive guide, we'll explore what the revolve command does, how to use it step-by-step, common mistakes to avoid, and practical tips for getting the most out of this feature.

## What Does the Revolve Command Do in Fusion 360?

The revolve command in Fusion 360 enables users to create 3D objects by rotating a 2D sketch around an axis. Think of it as wrapping a flat shape around a central line—imagine spinning a rectangle around its long edge to form a cylinder. This powerful modeling technique is ideal for producing symmetrical objects with rotational features, such as vases, cams, or hollow tubes.

Primarily, the revolve command transforms simple sketches into complex, smooth, and symmetrical parts with minimal effort, making it an essential tool for mechanical, product, and industrial designers. The key is that the shape is created by revolving a closed or open sketch profile, and the extent of rotation can be customized based on design requirements.

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

Using the revolve command involves a series of straightforward steps. Let's walk through the process with practical examples.

### 1. Prepare Your Sketch

- Choose or create a 2D sketch on a plane that contains the profile you want to revolve.
- The sketch should include the shape or profile you intend to revolve. It can be a closed loop (like a semi-circle or rectangle) or an open profile (like a line or arc).

## 2. Select the Revolve Tool

- Go to the toolbar and click on the "Create" dropdown menu.
- Locate and select the "Revolve" option. Alternatively, press the shortcut key "REVOLVE" if assigning custom hotkeys.

## 3. Select Your Sketch Profile

- Click on the sketch profile or profiles you wish to revolve.
- Ensure that only the desired geometry is selected for the revolution to avoid unwanted features.

## 4. Define the Axis of Revolution

- Click on an existing line or edge within the sketch that will act as your axis.
- If no suitable line exists, you can draw a new one in the sketch before selecting the revolve.

## 5. Set the Revolve Angle

- Input the rotation angle in degrees:
- For a full revolution, enter 360°.
- For a partial or semi-revolution, input a smaller angle, like 180°.
- You can also choose "To Object" or "Two Planes" options for more complex revolutions.

## 6. Adjust Additional Settings

- Select whether the revolution creates a solid or a surface.
- Use the "Operation" options to join, cut, or intersect with existing bodies.
- For advanced shaping, experiment with the "Tweak" setting for smooth transitions.

## 7. Click OK

- Confirm your settings by clicking "OK."
- Fusion 360 will generate the 3D model based on your defined parameters.

## Practical Example: Creating a Hollow Cylinder

- Sketch a circle on the XY plane representing the cross-section.
- Draw a line representing the axis of revolution.
- Select the circle as the profile, the line as the axis, and set the angle to 360°.
- Click OK to generate the hollow cylinder.

## Common Mistakes When Using the Revolve Command and How to Avoid Them

Even experienced designers occasionally make mistakes with the revolve feature. Here are common issues and solutions:

### 1. Forgetting to Select the Correct Axis

- Mistake: Revolving around an unintended line results in incorrect geometry.
- Solution: Always double-check the axis selection and ensure it's aligned with your intended rotation.

### 2. Using an Open Profile When a Closed Profile Is Needed

- Mistake: Open profiles may result in incomplete or unintended geometry.
- Solution: Verify whether your shape is closed or open, and adjust your sketch accordingly.

### 3. Not Fully Constraining the Sketch

- Mistake: Unconstrained sketches can cause unpredictable revolutions.
- Solution: Fully constrain your sketch to prevent accidental movement or scaling.

## 4. Ignoring the Partial Revolution Settings

- Mistake: Misunderstanding the "Angle" input, leading to unwanted features.
- Solution: Carefully input the desired degrees for the revolution and preview before finalizing.

## 5. Overlooking the "Operation" Settings

- Mistake: Not choosing the correct operation (Join, Cut, or Intersect) can lead to modeling errors.
- Solution: Understand what each operation does and select appropriately based on your design intent.

# Tips and Best Practices for Using the Revolve Command in Fusion 360

- Use construction lines to define axes for precise revolutions.
- When creating hollow shapes, combine the revolve with the shell or hollow command.
- Use the "Preview" feature to see the result before confirming the revolve.
- For complex shapes, consider combining multiple revolved and extruded features.
- Use the timeline to adjust the revolve parameters after initial creation.

## Practical Applications of the Revolve Command

The revolve function is

- Ideal for designing rotationally symmetric objects like:

- Pipes, tubes, and cylinders
- Mechanical gears and cams
- Vases and bottles
- Customized handles and knobs
- Rotational parts in assemblies

Understanding how to leverage the revolve command allows for efficient modeling and prototyping of many essential parts and products.

## Comparing the Revolve Command with Other Fusion 360 Features

<b>Feature</b>	<b>Use Case</b>	<b>Key Difference</b>
Revolve	Creating symmetrical 3D objects by rotation	Rotates a 2D profile around an axis

Extrude	Extending a 2D shape linearly	Adds depth or height without rotation
Sweep	Creating path-based shapes	Follows a specified path for complex profiles
Loft	Blending between multiple profiles	Creates smooth transitions between different shapes

While each tool has its purpose, the revolve command excels at producing symmetrical forms with minimal effort.

## Conclusion

The revolve command in Fusion 360 is a fundamental feature that allows you to efficiently transform 2D sketches into fully three-dimensional, rotationally symmetric models. By mastering this tool, you can streamline your design workflow and create complex parts with precision and

ease. Remember to carefully select your sketch profile and axis, verify the revolution angle, and utilize best practices to avoid common pitfalls. Whether you're designing a simple cylinder or an intricate cam mechanism, the revolve command is a versatile and essential skill for any Fusion 360 user.

## FAQ

### 1. What is the primary function of the revolve command in Fusion 360?

**Ans :** It creates 3D objects by rotating a 2D sketch around an axis.

### 2. Can I use the revolve command for creating hollow objects?

**Ans :** Yes, by combining the revolve with shell or hollow features.

### 3. How do I create a partial rotation using the revolve command?

**Ans :** Enter the desired angle less than 360° in the angle input field.

### 4. What should I do if my revolve doesn't produce the expected shape?

**Ans :** Check that your sketch profile is correctly constrained and that the axis is properly selected.

### 5. Is it possible to revolve multiple sketches at once?

**Ans :** No, Fusion 360 typically requires you to select and revolve one profile at a time, but multiple profiles on the same axis can sometimes be revolved together.

### 6. How can I modify the revolve after creating it?

**Ans :** Edit the feature in the timeline or parameters to adjust the profile, axis, or angle.

### 7. Can I use the revolve command with open profiles?

**Ans :** Yes, but the results depend on the geometry; open profiles may produce surfaces or incomplete solids.

# 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

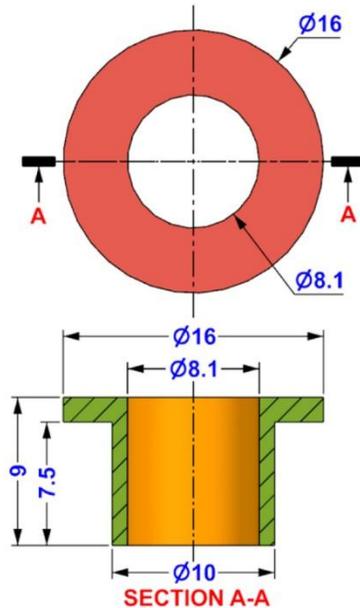
## 500+ PRACTICE EXERCISES

### • Sketching



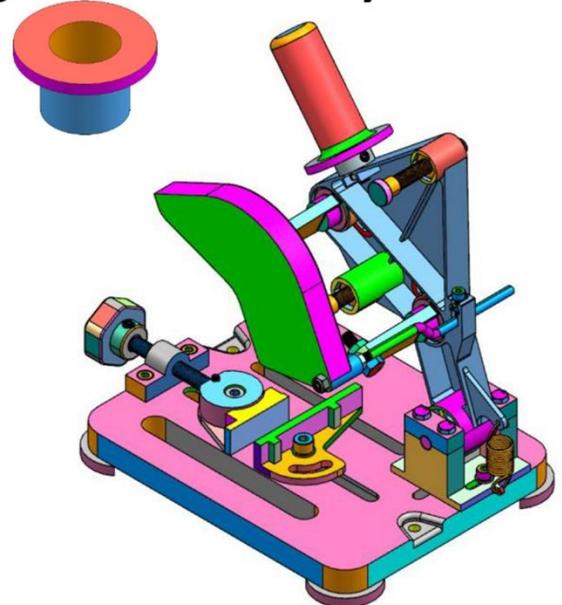
2D Sketching

### • 3D Modeling



3D Modeling

### • Assembly



Assembly

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](mailto: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](http://cadin360.com)

Copyright © 2025 by CADIN360, All rights reserved.

This book is copyrighted and the CADIN360 reserves all rights.

No part of this publication may be reproduced, stored in a retrieval system or transmitted, transcribed, stored in retrieval system or translated into any language, in any form or by any means, electronic, mechanical, photocopying, recording, scanning or otherwise, without the prior written permission of the publisher & Author.

## **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](mailto: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.

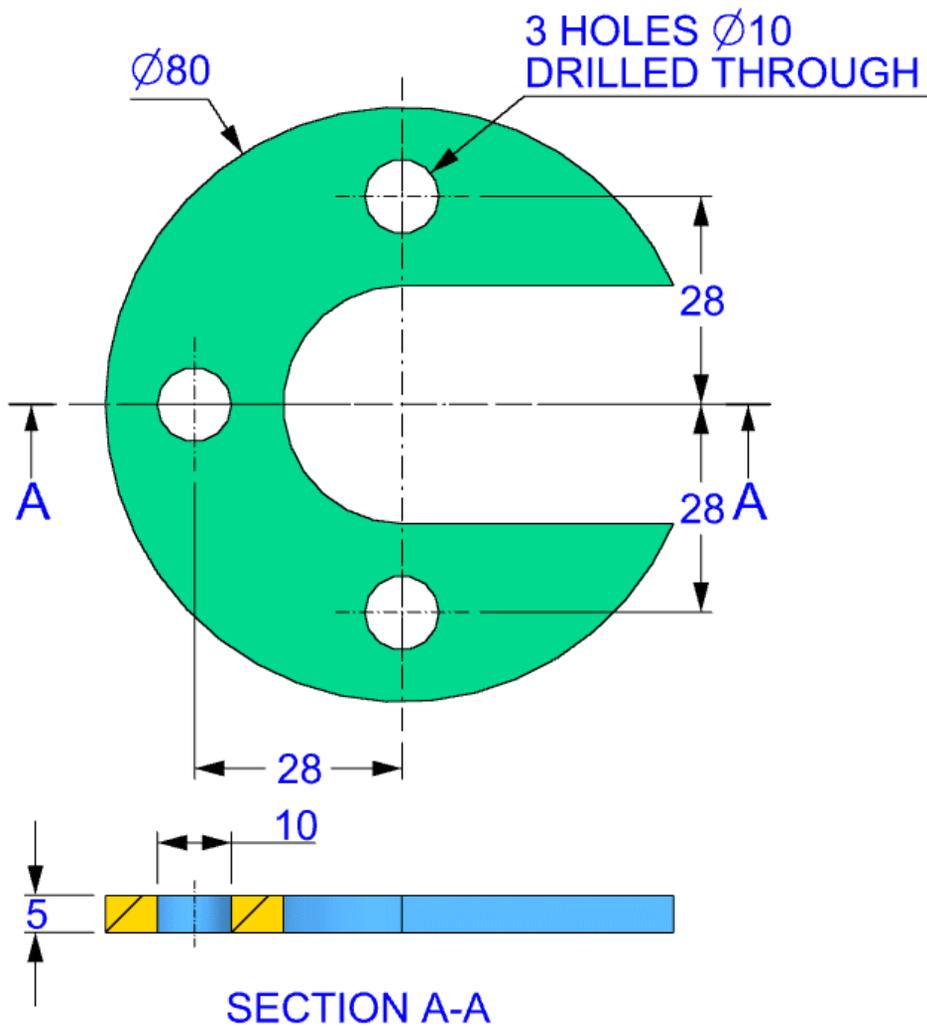
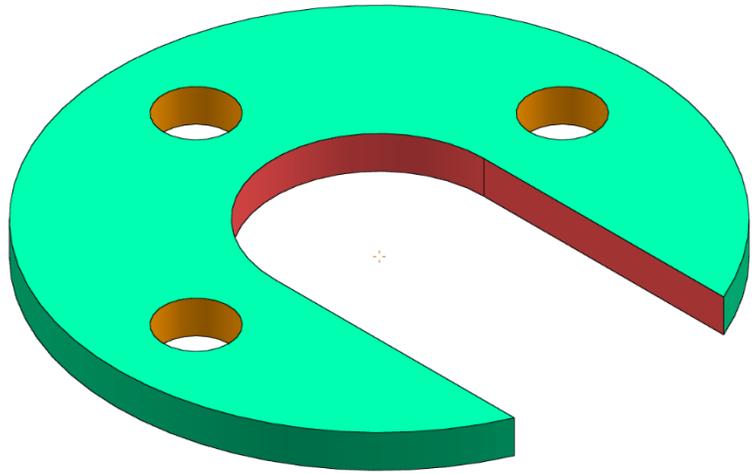
### **Note:**

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

Happy learning!  
– Team CADIN360

3D

## 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

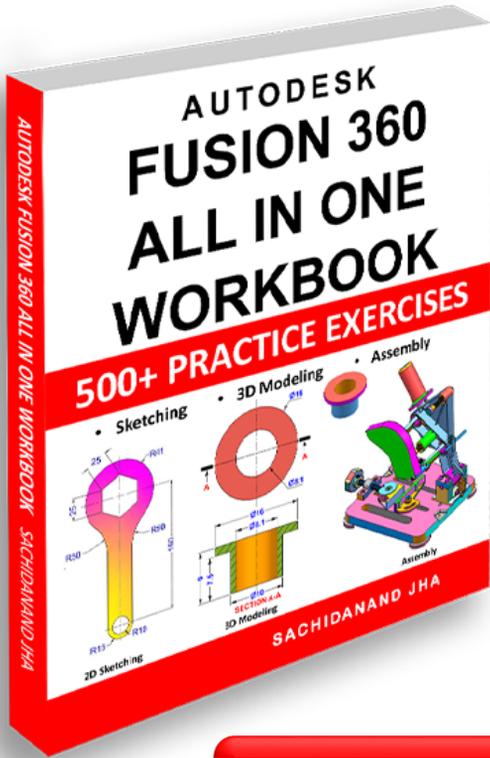
- A software-specific complete sample PDF
- Multiple real practice exercises (not a single file)
- Same quality as our professional training material
- Compatible with the latest software version

## 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](http://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](mailto:cadin360@gmail.com)

🌐 Website: [www.cadin360.com](http://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 3<sup>rd</sup> Angle projection
- All dimensions are in mm(metric system)