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 EXTRUDE IN OPPOSITE DIRECTION IN FUSION 360

• LEARN • • APPLY • • GROW •

Introduction

Extruding in Fusion 360 is a foundational technique for creating 3D models from sketches and designs. Typically, users extrude in a single direction, either outward or inward from the sketch plane. However, there are many practical scenarios where extruding in opposite directions in Fusion 360 is essential. For instance, when designing complex assemblies, creating symmetrical parts, or adding features that extend both ways from a central plane, understanding how to extrude in opposite directions becomes crucial.

This guide will walk you through the step-by-step process of performing an extrusion in opposite directions in Fusion 360. You'll learn how to set up your sketches, adjust extrusion settings for bidirectional operation, and troubleshoot common issues. Whether you're a beginner or someone looking to refine your Fusion 360 skills, this comprehensive tutorial will give you the practical knowledge to innovate with confidence.

How to Extrude in Opposite Direction in Fusion 360

Performing an extrusion in both directions in Fusion 360 is straightforward once you understand the available options. Here, we'll explore the methods to achieve this, along with tips to ensure a smooth workflow.

1. Preparing Your Sketch and Design

Before starting the extrusion, ensure your sketch is properly prepared:

- Create a 2D sketch on the plane where you want to begin your extrusion.
- Clearly define the shape, dimensions, and constraints.
- For symmetrical or opposite-direction extrusions, consider sketching the features centrally or on opposite sides.

2. Launching the Extrude Tool

To start extruding:

- Select the profile you wish to extrude.
- Click on the "Create" menu in the toolbar.
- Choose **"Extrude"** from the dropdown, or simply press **shortcut key E**.

3. Modifying the Extrude Direction

Once the extrusion dialog box appears:

- Under **Direction**, you typically see options like "One Side," "Two Sides," or "Symmetric."
- To extrude in opposite directions:
- **Choose "Two Sides"**: This allows you to specify different distances for each side.
- **Choose "Symmetric"**: This creates an equal extrude in both directions from the sketch plane.

4. Setting Distances for Opposite Extrusions

- In the dialog box, you'll see input fields for each side of the extrusion.
- Enter the desired length for each side, allowing for different extents in opposite directions.
- For example, set 5mm on one side and 10mm on the other for asymmetric opposite extrusions.

5. Confirm and Finalize

- After setting the distances:
- Click **OK** to complete the extrusion.
- Inspect the model to ensure the extrusion follows your intended directions.

Practical Examples of Extruding in Opposite Directions

Understanding how to extrude in opposite directions becomes especially powerful when applied to real-world projects.

Example 1: Creating Symmetrical Parts

Suppose you're designing a custom bracket that extends equally on both sides of a mounting surface:

- Use the "Symmetric" option.
- Input the desired total height, and Fusion 360 will split it equally in both directions.

Example 2: Multi-Feature Assembly

When adding features such as bosses or ribs that extend in both directions from a thin wall:

- Select "Two Sides."
- Specify different dimensions per side depending on design needs.

Example 3: Complex Cutouts and Shapes

For creating cutouts or features that extend in both directions:

- Use the "Two Sides" or "Symmetric" options to control the feature extension precisely.

Common Mistakes and How to Avoid Them

Even experienced users can stumble occasionally. Here are common mistakes and ways to avoid them:

1. Forgetting to Select "Two Sides" or "Symmetric"

- **Solution:** Always double-check your extrusion type before inputting distances.

2. Not Adjusting for Fused Geometry

- Fused or overlapping profiles may cause errors during opposite extrusions.
- **Solution:** Repair or simplify your sketch before extruding.

3. Inputting Incorrect Distances

- Entering sizes that don't match the design intent.
- **Solution:** Carefully plan your distances, or use dimensions and constraints to automate this.

4. Not Checking the Direction

- Over-extruding in unintended directions can lead to flawed models.
- **Solution:** Visualize your extrusion in the preview window and adjust accordingly.

Pro Tips for Mastering Opposite Direction Extrusions

- Always use the **"Two Sides"** or **"Symmetric"** options for precise control.
- Use **Constraints and Dimensions** in sketches to make your extrusion parameters flexible.
- When designing symmetric components, consider creating a **centerline sketch** to facilitate symmetrical extrusion.
- Experiment with **extents and distances** in the extrusion dialog to understand how they influence the finished part.

- Save your work regularly and consider creating **parameters** to control your extrusion dimensions for easy adjustments later.

Comparison: One-Directional vs. Opposite (Two-Sides or Symmetric) Extrusion

Feature	One-Directional Extrusion	Opposite Direction (Two Sides / Symmetric)
Use Case	Extending material in one direction only	Extending material in both directions from the sketch plane
Control	Single distance input	Two distances or a symmetric total

Ideal for	Simple parts, side-specific features	Symmetrical and bi-directional features
Setup Complexity	Minimal	Slightly more setup, but more flexible

Using the appropriate options allows for smarter, more precise modeling—especially useful for complex geometries requiring balanced or asymmetrical extensions.

Conclusion

Extruding in opposite directions in Fusion 360 enhances your modeling capabilities, allowing for more precise, symmetrical, and complex designs. Whether you're creating parts that extend equally or asymmetrically from a sketch plane, understanding the "Two Sides" and "Symmetric" options is key.

Practicing with these tools and techniques empowers you to design more efficiently and accurately. By mastering opposite direction extrusions, you'll unlock new possibilities for your CAD projects—culminating in more professional and refined results.

FAQ

1. How do I extrude in both directions in Fusion 360?

Ans: Select the "Two Sides" or "Symmetric" option in the extrude dialog box and specify distances for each side.

2. What is the difference between "Two Sides" and "Symmetric" in Fusion 360?

Ans: "Two Sides" allows you to specify different distances for each side, while "Symmetric" splits the total extrude equally from the center plane.

3. Can I change the direction of an extrusion after creating it?

Ans: Yes, you can edit the extrusion feature and modify the direction and distances in the feature's dialog box or timeline.

4. How do I ensure my sketch is suitable for opposite extrusion?

Ans: Use constraints and symmetry lines within your sketch to facilitate balanced and accurate extrusions in both directions.

5. Can I extrude in both directions without using the "Two Sides" option?

Ans: No, for true opposite directions in a single operation, the "Two Sides" or "Symmetric" options are required; otherwise, perform separate extrusions.

6. How do I troubleshoot errors when extruding in opposite directions?

Ans: Check for overlapping or fused profiles, ensure you have selected the correct extrusion type, and verify your sketch constraints for accuracy.

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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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

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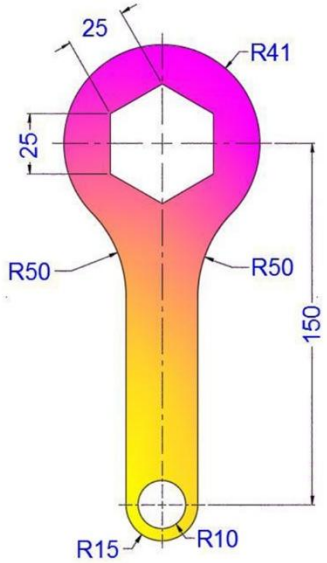
- 3D Modeling
- Assembly



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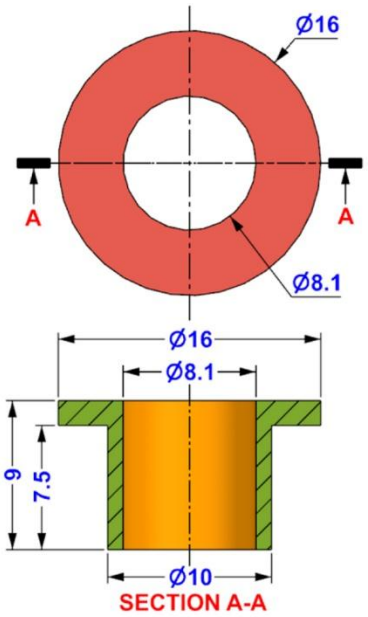
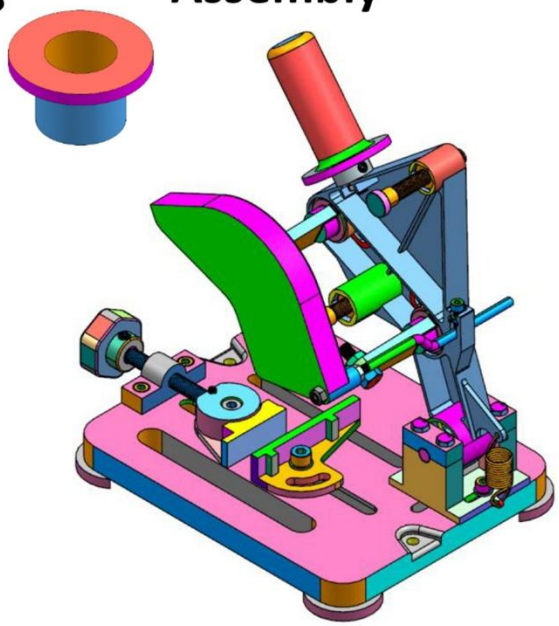


3D Modeling



Assembly

SACHIDANAND JHA

AUTODESK

FUSION 360 ALL IN ONE WORKBOOK

500+ PRACTICE EXERCISES

2D Sketching • 3D Modeling • Assembly Drawings

SACHIDANAND JHA

Dear Reader,

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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.

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

- 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:

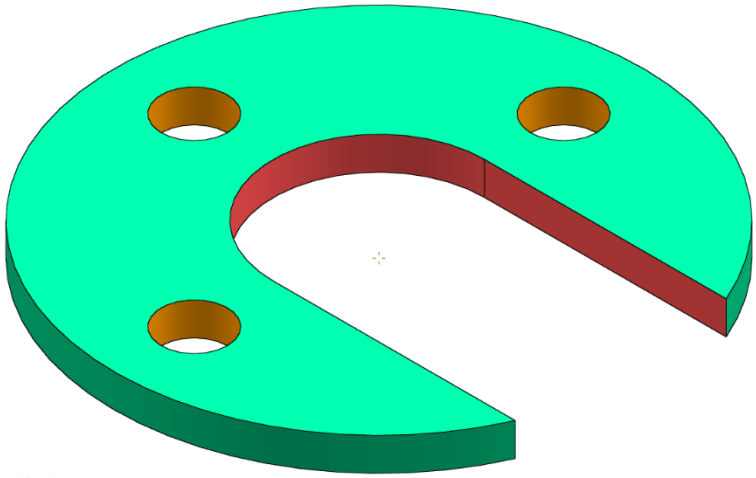
- Complete the exercises in order, or jump to any skill level you prefer.
- All dimensions are in millimeters.
- Where dimensions are missing, apply logic or practice estimation.
- This book is ideal for both students and professionals preparing for industry design work.

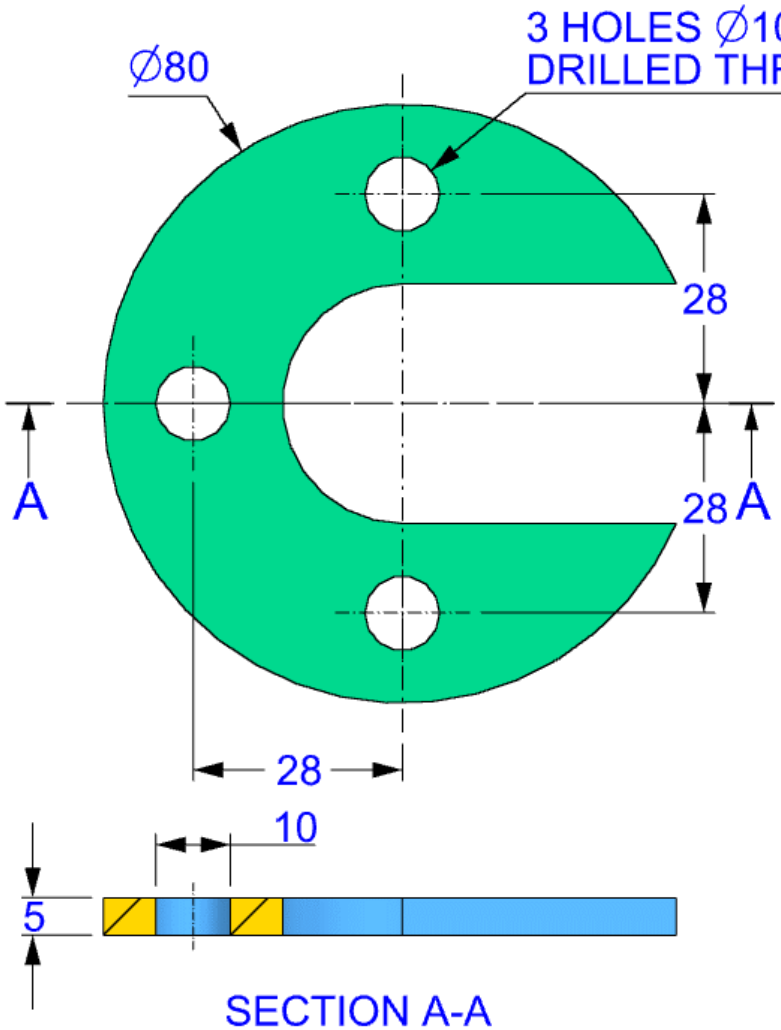
Note:

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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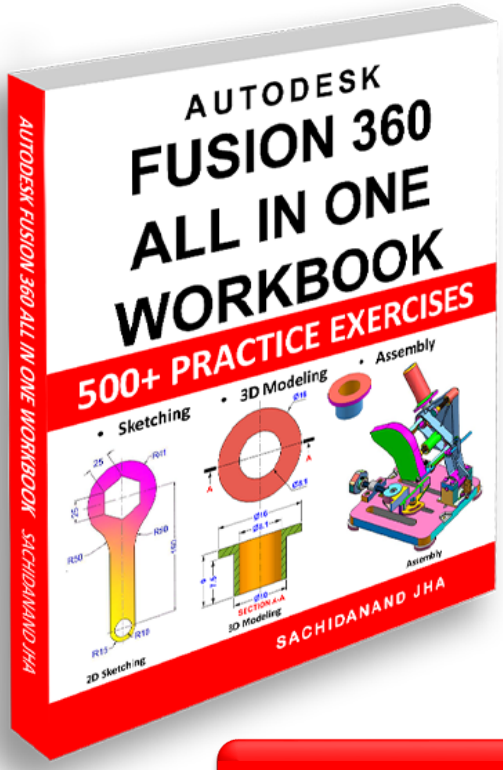

What's Included in the FUSION 360 ALL IN ONE WORKBOOK?

- ✓ 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
- ✓ Get All Assembly Exercises in .STP file
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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)