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 EQUALLY ON BOTH SIDES IN FUSION 360

• LEARN • • APPLY • • GROW •

Introduction

Extruding equally on both sides in Fusion 360 is a common task for designers aiming for symmetrical features, whether it’s creating balanced ridges, slots, or complex geometries. Achieving precision in these extrusions ensures that your models are both functional and aesthetically pleasing. This tutorial provides a detailed, step-by-step guide to help you extrude equally on both sides of a sketch, along with tips, common mistakes, and practical examples. Whether you're a beginner or looking to refine your workflow, mastering this technique will significantly enhance your proficiency in Fusion 360.

Understanding the Basics of Extrusion in Fusion 360

Before diving into how to extrude equally on both sides, it’s crucial to understand the general extrusion tools available:

- **Single-sided extrusion:** Extends a sketch profile in one direction.
- **Symmetric extrusion:** Extends equally on both sides, central to achieving balanced features.
- **One-side extrusions with Distance or To Object options:** Custom control over extrusion direction and length.

Fusion 360 offers several options to manipulate how a profile is extruded; selecting the right method simplifies symmetrical modeling.

Step-by-Step Guide to Extruding Equally on Both Sides

To ensure an extrusion occurs equally on both sides, follow these precise steps:

1. Prepare Your Sketch

- Begin with a clean, fully constrained sketch.
- Draw the profile you wish to extrude, including any internal or external features.

- Ensure your sketch is closed; open profiles cannot be extruded properly.

2. Open the Extrude Tool

- After completing your sketch, switch to the **Solid** tab.
- Click on **Create** > **Extrude** or press the shortcut key **E**.

3. Select the Profile for Extrusion

- Click inside the sketch profile to select it.
- Review the preview to verify the selected area.

4. Choose the Extrude Direction and Distance

- In the **Extrude** dialog box, locate the **Direction** options.
- Select **Symmetric** from the dropdown menu.
- Input the total extrusion distance; Fusion 360 will automatically split this equally on both sides.

5. Set the Extrusion Distance

- Enter the total desired length (e.g., 10 mm).
- Fusion 360 will extrude 5 mm in one direction and 5 mm in the opposite, ensuring symmetry.

6. Confirm the Operation

- Click **OK** to execute the symmetric extrusion.
- The feature should be perfectly balanced on both sides of your sketch plane.

7. Verify and Adjust if Needed

- Check the extrusion for accuracy.
- If adjustments are necessary, double-click the feature in the timeline, modify the distance, and reapply.

Practical Examples of Equally Extruded Features

Example 1: Creating a Balanced Groove

Suppose you want to create a groove centered on a face. Drawing a rectangular profile and extruding symmetrically ensures the groove is centered and evenly spaced from the edges.

Example 2: Symmetric Ridges on a Panel

Designing a panel with evenly spaced ridges involves sketching the profile of each ridge and applying symmetric extrusion, maintaining uniformity across the surface.

Example 3: Mirrored Features for Mechanical Parts

When designing parts that require mirrored features, extruding symmetrically simplifies the process, ensuring both sides match perfectly without additional mirroring steps.

Common Mistakes and How to Avoid Them

- **Not selecting “Symmetric” in the extrude options:** This misses the goal of equal extrusion on both sides.
- **Using a fixed distance instead of symmetric:** Leads to unbalanced features.
- **Sketch not being fully constrained or open profiles:** Causes unpredictable extrusion results.
- **Forgetting to set the correct direction:** Sometimes default is set to one side; double-check the options.
- **Skipping the preview step:** Always verify the preview before confirming.

Pro Tips and Best Practices for Symmetrical Extrusions

- Always use the **Symmetric** option when the goal is to create balanced features.
- Assign a neutral sketch plane (e.g., XY plane) as your centerline for easier symmetry.
- Utilize construction lines in sketches to define the center of symmetric features.
- Combine symmetric extrusion with mirror features for complex symmetrical designs.
- Use parameter-driven dimensions to easily modify the total extrude length that automatically updates on both sides.
- Organize your timeline and feature order for easy edits.

Comparing Symmetric vs. Asymmetric Extrusions

Feature	Symmetric Extrusion	Asymmetric Extrusion
Purpose	Creates features equally on both sides of the sketch plane	Extends in one direction only

Use case	Centered features, balanced designs	When a feature needs to extend in a specific direction
Setup complexity	Simple; just select “Symmetric” in the extrude dialog	May require manual input and adjustments
Editing flexibility	Easy to modify total distance, maintained symmetry	Adjustment may break symmetry

Using the correct extrusion method based on your design intent improves workflow efficiency and ensures precise results.

Conclusion

Mastering how to extrude equally on both sides in Fusion 360 significantly enhances your modeling capabilities, enabling you to create symmetrical, balanced features with ease. By

following the step-by-step instructions, avoiding common mistakes, and applying best practices, you can streamline your design process and produce professional-quality models. Whether designing mechanical parts, aesthetic features, or complex assemblies, understanding symmetric extrusion is an essential skill for every Fusion 360 user.

FAQ

1. How do I extrude equally on both sides in Fusion 360?

Ans: Select the **Symmetric** option in the Extrude dialog box and input the total extrude distance; Fusion 360 will split it evenly on both sides.

2. Can I change a one-sided extrude to a symmetric one after creating it?

Ans: Yes, double-click the extrude feature in the timeline, select **Edit Feature**, and then choose the **Symmetric** option.

3. What should I do if the symmetric extrusion isn't balanced?

Ans: Ensure you have selected **Symmetric** in the extrusion options, and verify your sketch is properly constrained and centered.

4. How do I create a centered feature without using the symmetric extrude?

Ans: Draw a centerline, constrain your sketch around it, and extrude in one direction equally in both length, or mirror the features afterward.

5. Is it possible to extrude symmetrically in complex patterns?

Ans: Yes, after performing a symmetric extrude, you can pattern or mirror features to create complex symmetrical designs efficiently.

6. What are some tips for ensuring perfect symmetry in Fusion 360?

Ans: Use construction lines, centerpoints, and a dedicated plane as your symmetry reference to maintain precision.

7. Can I extrude symmetrically along curved surfaces?

Ans: Symmetric extrusions are primarily for planar profiles; for curved surfaces, other tools like sweeps or lofts are more appropriate.

This guide aims to help you achieve precise, symmetric extrusions quickly and confidently, streamlining your Fusion 360 modeling workflow.

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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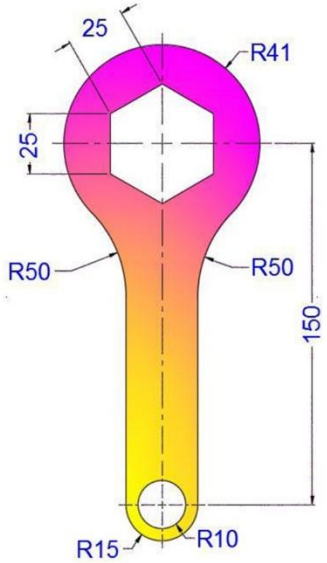
- Assembly



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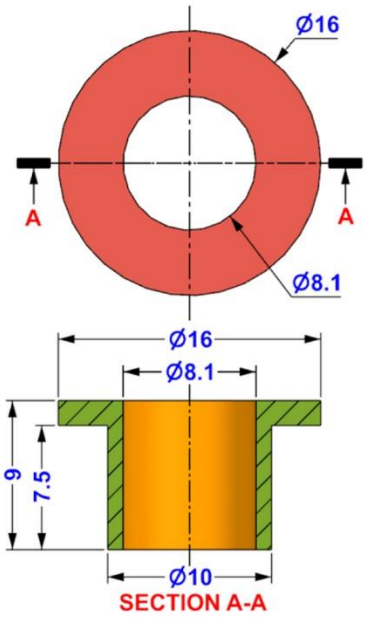

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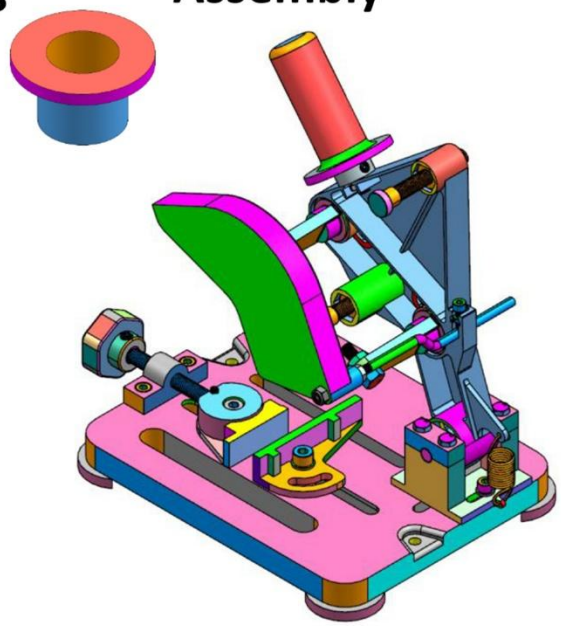

Assembly

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

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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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.

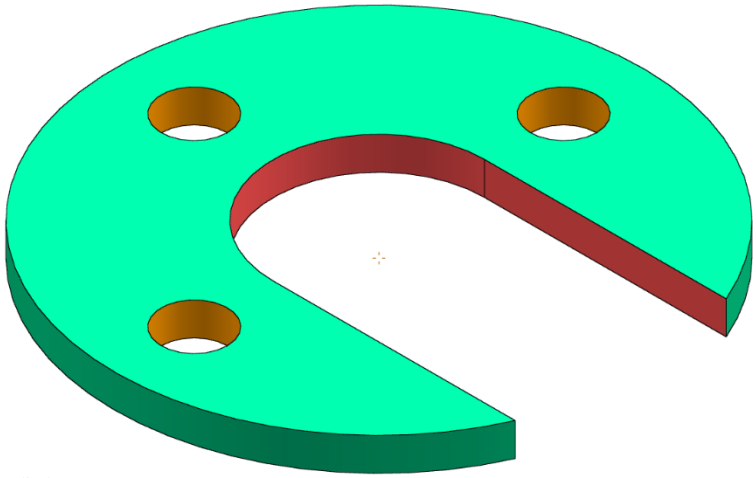
Tips for Best Use:

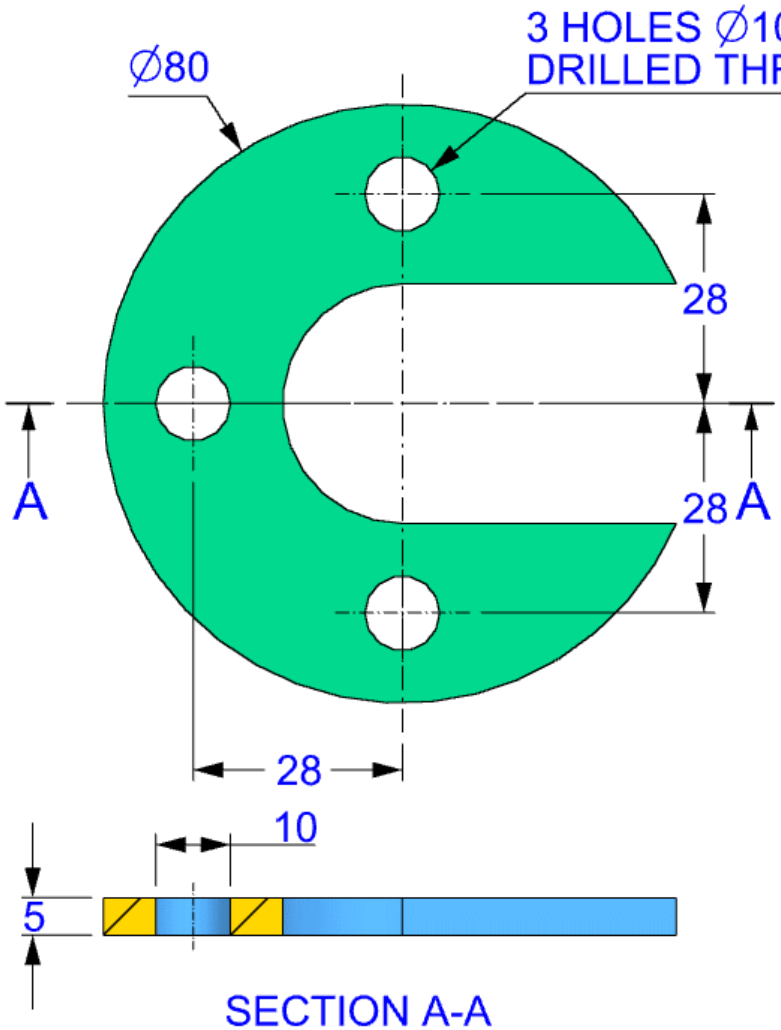
- Complete the exercises in order, or jump to any skill level you prefer.
- All dimensions are in millimeters.
- Where dimensions are missing, apply logic or practice estimation.
- This book is ideal for both students and professionals preparing for industry design work.

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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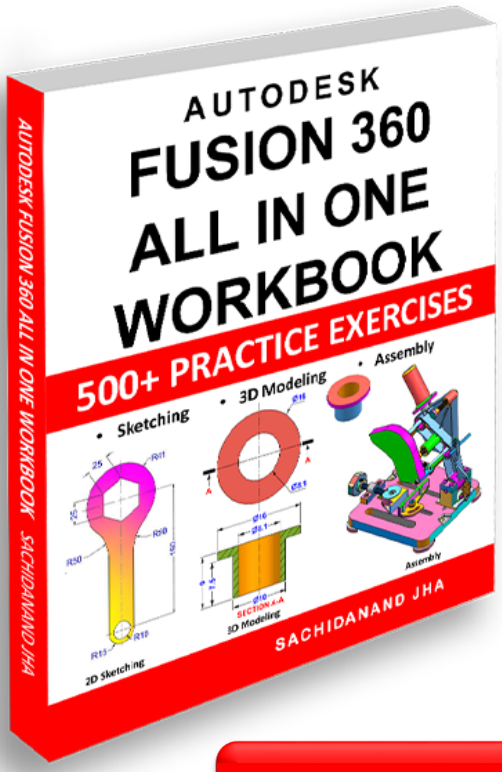

What's Included in the FUSION 360 ALL IN ONE WORKBOOK?

- ✓ 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)