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.

Disclaimer:

— CADIN360 Team

HOW TO CHANGE EXTRUSION LATER IN FUSION 360

• LEARN • • APPLY • • GROW •

Introduction

Changing the extrusion later in Fusion 360 is a common task for designers looking to modify their 3D models after initial creation. Whether you need to tweak a shape, correct an error, or refine your design, understanding how to adjust extrusions efficiently is vital. This guide provides a detailed, step-by-step process to help you learn how to change extrusion later in Fusion 360. By mastering these techniques, you'll enhance your modeling flexibility, save time, and produce more accurate and polished designs.

Understanding Fusion 360's Extrusion Tool

Before diving into how to change extrusions later, it’s important to understand what extrusion means in Fusion 360. Extrusion involves creating a 3D shape by extending a 2D sketch along a specific axis. When you extrude, you turn flat sketches into three-dimensional models.

Key aspects of extrusion in Fusion 360:

- Material thickness
- Direction (one side, two sides, symmetric)
- Whether the extrusion is a new body or adds to an existing one
- The ability to modify the extrusion parameters after creation

Fusion 360 uses a parametric modeling approach, allowing users to revisit and alter earlier steps, like extrusion, quickly.

How to Change Extrusion Later in Fusion 360: Step-by-Step Guide

Changing an extrusion after creating it involves editing the feature associated with that extrusion. Here’s how to do it:

1. Open Your Fusion 360 Project

- Launch Fusion 360 and open the relevant design file.
- Locate the body or component containing the extrusion you wish to modify.

2. Access the Timeline at the Bottom

- Fuse 360 tracks your modeling history in the timeline bar located at the bottom of the workspace.
- Find the extrusion feature, which is usually labeled with the command used, like "Extrude."

3. Find the Extrude Feature

- Scroll through the timeline to locate the extrusion.
- If you named it during creation, it will be easier to identify.

4. Edit the Extrude Feature

- Right-click on the extrusion feature.
- Select **Edit Feature** from the context menu.

5. Adjust the Extrusion Parameters

- The **Extrude** dialog box will open, showing options such as:
- Distance or extent of extrusion
- Direction (Symmetric, One Side, Two Sides)
- Operation type (Join, Cut, New Body)
- Taper angle (if applicable)
- Modify these parameters as needed:
- Change the distance value to increase or decrease extrusion length.
- Switch direction or operation type to alter how the shape interacts with other features.

6. Confirm the Changes

- After adjustments, click **OK**.
- Fusion 360 will update the model accordingly, reflecting your new extrusion parameters.

7. Verify and Fine-Tune

- Inspect your model for accuracy.
- Make further adjustments if necessary by repeating the editing process.

Practical Examples of Changing Extrusions in Different Scenarios

Example 1: Extending an Existing Part

Suppose you initially extruded a rectangle to create a block. Later, you realize you need it to be longer:

- Follow the steps above to edit the extrusion.
- Increase the distance value.
- Review the change in your model workspace.
- Save the project.

Example 2: Reducing or Removing an Extrusion

If the extrusion is too long or you want to undo it:

- Access the **Edit Feature**.
- Reduce the distance to zero to remove the extrusion.

- Alternatively, delete the feature from the timeline and redraw with modified parameters.

Example 3: Changing the Direction or Operation

Suppose you initially extruded inward but need to extrude outward:

- Edit the existing extrusion.
- Change the direction setting from **One Side** to **Symmetric** or adjust the arrow in the dialog box.
- Confirm the change to see the new shape.

Common Mistakes When Changing Extrusion Later

- **Modifying the wrong feature:** Always double-check the feature in the timeline to ensure you're editing the intended extrusion.
- **Ignoring dependencies:** Changes might affect subsequent features. Be cautious if other features depend on the extrusion.
- **Accidental deletion:** Deleting an extrusion instead of editing can cause loss of features. Use the right-click **Edit Feature** method.
- **Forgetting to update sketches:** If your extrusion is based on a sketch which has changed or been deleted, you may need to edit the sketch first.

Best Practices for Adjusting Extrusions

- **Name your features:** When creating extrusions, give them descriptive names to easily find and edit later.
- **Use parameters:** Define parameters (like length, width) for easy global adjustments in future revisions.

- **Plan your workflow:** Build your model in a way that allows easy modification, keeping your timeline organized.

Comparison: Editing an Extrusion vs Creating a New One

Aspect	Editing an Existing Extrusion	Creating a New Extrusion
Time required	Quicker	More time-consuming
Impact on design history	Preserves previous steps	Adds a new feature to timeline

Flexibility for revisions	High	Depends on sketch/current state
Risk of breaking dependencies	Lower if carefully edited	Higher if not integrated properly

Understanding this distinction helps in managing your model systematically.

Conclusion

Learning how to change extrusion later in Fusion 360 is fundamental to effective 3D modeling. By leveraging the model history and editing features, you can make precise adjustments that refine your design without starting from scratch. Remember to access the timeline, right-click the extrusion feature, and tweak the parameters to your liking. Adopting best practices like naming your features and organizing your workflow will make future modifications even easier, boosting your productivity and confidence with Fusion 360.

FAQ

1. How do I modify an extrusion without affecting other features in Fusion 360?

Ans: You can right-click the extrusion in the timeline and select "Edit Feature" to modify it without impacting other features.

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

Ans: Yes, by editing the extrusion feature, you can change its direction setting, such as from one side to symmetric or two sides.

3. What should I do if my changes to an extrusion don't update the model?

Ans: Ensure you clicked "OK" after editing and that the feature is not hidden or suppressed; refresh or rebuild the model if necessary.

4. Is it possible to change an extrusion's operation from "Join" to "Cut" later?

Ans: Yes, by editing the extrusion feature, you can switch the operation type from "Join" to "Cut" to modify how it interacts with other bodies.

5. How can I delete an extrusion without affecting the rest of my model?

Ans: Right-click on the extrusion in the timeline and select "Delete" to remove it; be aware that this may affect dependent features.

6. Can I revert an extrusion to its original dimensions after modification?

Ans: Yes, by editing the extrusion feature and restoring the previous parameter values, you can revert to the original dimensions.

7. What’s the best way to organize extrusions for easier editing in complex models?

Ans: Name each extrusion descriptively during creation and keep your timeline tidy to quickly locate features for later edits.

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

FUSION 360 ALL IN ONE WORKBOOK

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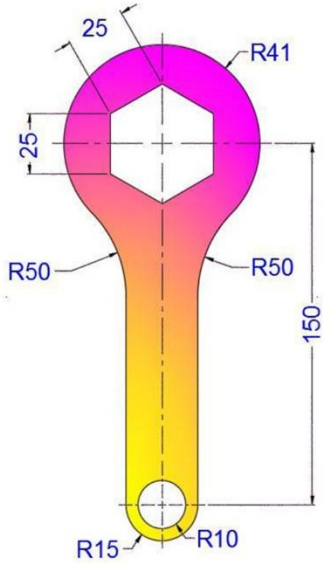
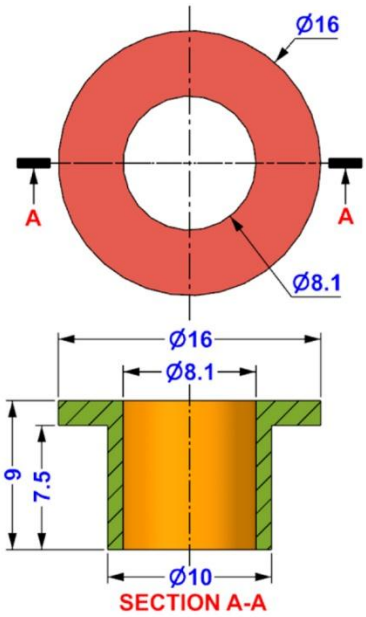
- Assembly



2D Sketching



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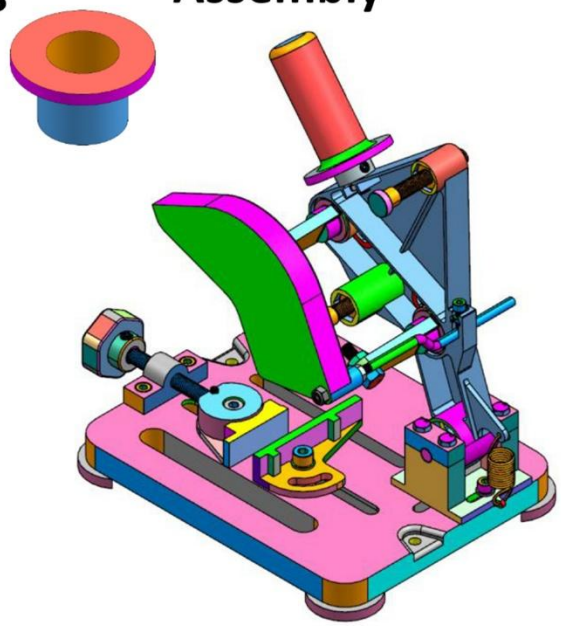

Assembly

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.

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.

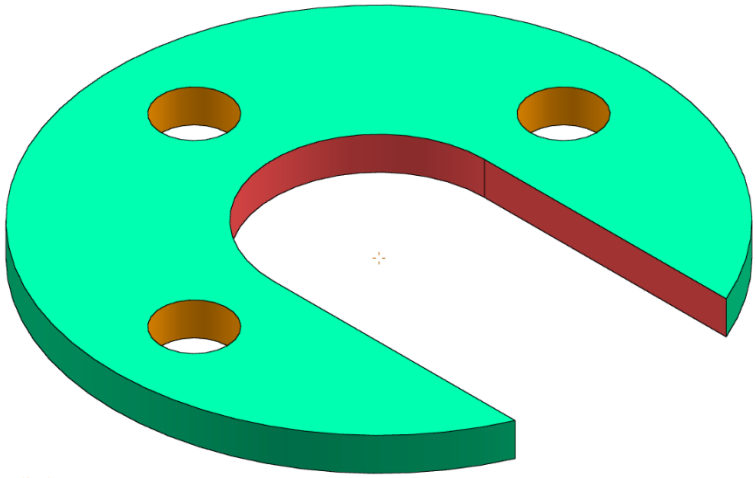
Tips for Best Use:

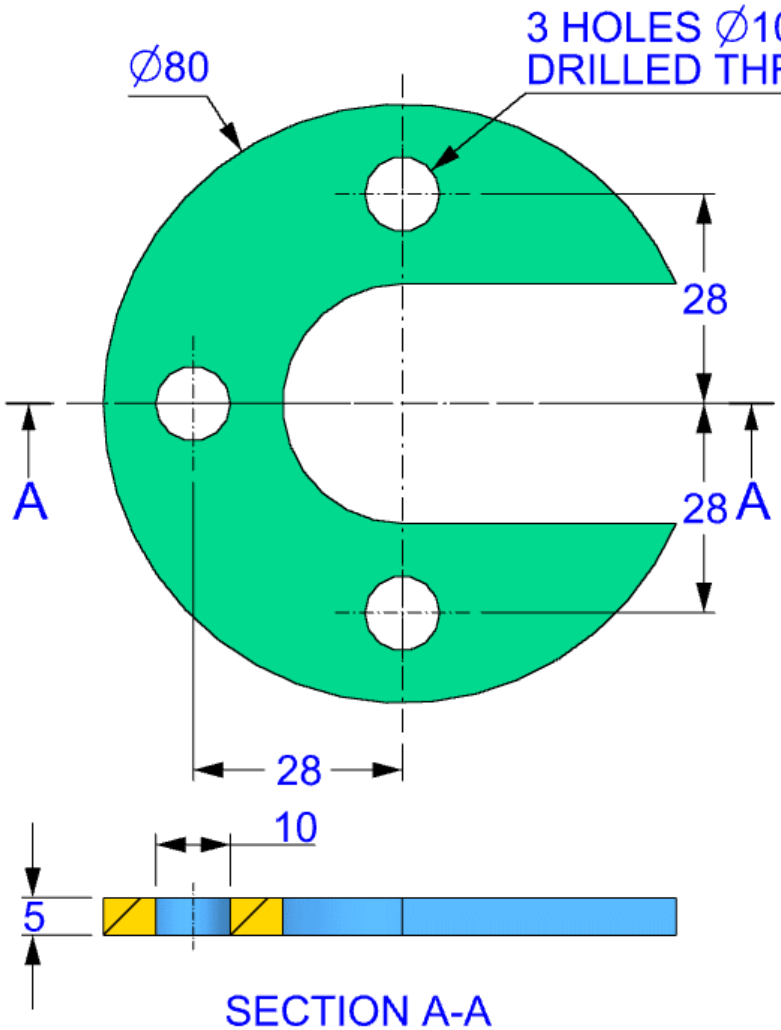
- Complete the exercises in order, or jump to any skill level you prefer.
- All dimensions are in millimeters.
- Where dimensions are missing, apply logic or practice estimation.
- This book is ideal for both students and professionals preparing for industry design work.

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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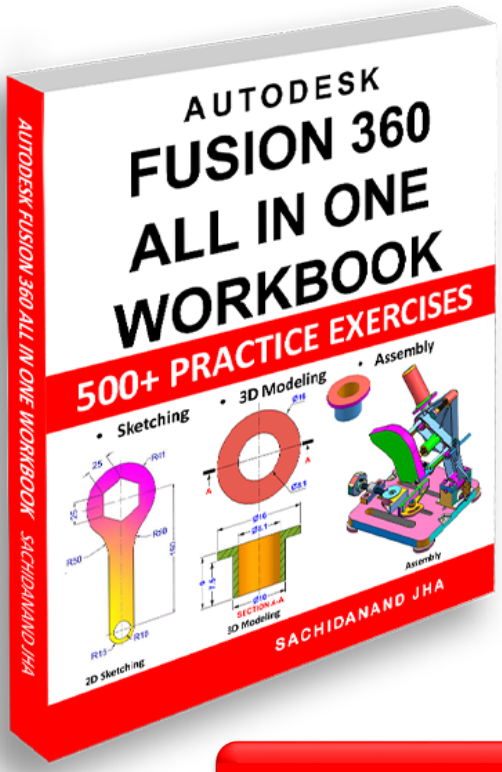

What's Included in the FUSION 360 ALL IN ONE WORKBOOK?

- ✓ 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)