

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



DIFFERENCE BETWEEN JOIN, CUT, AND NEW BODY IN FUSION 360

• LEARN • • APPLY • • GROW •

Introduction

Fusion 360, Autodesk's powerful 3D CAD, CAM, and CAE tool, offers a wide array of features to help designers and engineers create complex models with precision. Among its many tools, the "Join," "Cut," and "New Body" commands are fundamental for manipulating and combining bodies during the design process. Understanding the difference between Join, Cut, and New Body in Fusion 360 is crucial for working efficiently and achieving the desired design outcomes. In this in-depth guide, we'll explore each command's purpose, how and when to use them, and share practical examples for better clarity.

Understanding the Basic Functions: Join, Cut, and New Body

Fusion 360 provides a flexible set of tools to modify 3D models. Let's first break down each command's core purpose:

1. Join

The "Join" operation is used to merge two or more bodies into a single, unified body. This is especially useful when you want to create a seamless, continuous part without internal boundaries.

2. Cut

The "Cut" operation removes material from a body using another body or sketch as the cutting tool. It's ideal for creating holes, slots, or complex shapes by subtracting matter from an existing body.

3. New Body

Creating a "New Body" allows you to add a separate, independent component within the same design workspace. This is essential when designing assemblies or when multiple objects need to be manipulated separately.

Step-by-step Instructions for Using Join, Cut, and New Body in Fusion 360

1. Applying the Join Command

- Begin by selecting the "Modify" menu from the toolbar.
- Choose "Combine" from the dropdown options.
- In the "Combine" dialogue box:
- Set the "Operation" to **Join**.
- Select two or more bodies in the canvas that you want to merge.
- Confirm by clicking OK.

This operation will fuse all selected bodies into one continuous body.

Practical example: Combining two half-spheres to create a seamless orb.

2. Using the Cut Command

- Access the "Modify" menu.
- Select "Combine."
- In the "Combine" dialogue box:
- Set the "Operation" to **Cut**.
- Select the target body (the object you want to cut into).
- Choose the tool body (the object that acts as the cutting shape).
- Confirm with OK.

You can also perform cut operations using sketches with the "Extrude" feature set to "Cut," allowing precise control over the cut shape.

Practical example: Drilling a hole through a shaft using a cylinder as the cutting body.

3. Creating a New Body

- When creating or moving objects, ensure the "New Body" option is selected in the design pipeline.
- For example:
- When sketching a rectangle and extruding it, ensure the option "New Body" is checked in the Extrude dialog box if you want to keep this as an independent part.
- When importing parts, the software can automatically assign them as new bodies.

This method is vital for managing multiple components within an assembly or multi-body model.

Practical example: Creating separate components like a chassis and internal mounting brackets within the same Fusion 360 file.

Practical Applications and Examples

Combining Bodies with Join

Suppose you design two mechanical components, a gear and a shaft, and want them to act as a single piece in the assembly. Using the Join feature, you can fuse them into a solid body, ensuring smooth transfer of load during simulation analysis.

Using Cut for Subtractive Manufacturing

If you need to create a precisely located hole or notch—say for mounting purposes—you can use the Cut operation with a cylindrical or rectangular tool that intersects the body at the desired location, removing material where needed.

Keeping Parts Separate with New Body

When designing an assembly, such as a simple robot with motors, sensors, and structural components, creating each as a separate New Body allows for easier modifications, material assignment, and assembly constraints later.

Common Mistakes and How to Avoid Them

- **Accidentally merging bodies when you want to keep them separate:** Always verify the "Operation" in the Combine dialogue—use "New Body" for separation.
- **Not selecting the correct bodies for Cut:** Double-check the selection in the dialog box, especially when working with complex models.
- **Overusing Join when separate bodies are needed:** Use "New Body" or "Cut" instead of Join if you want to keep parts independent.

Pro tip: Use the "Bodies" folder in the browser to keep track of your design components and see how the commands affect each.

Best Practices and Tips

- Always organize your bodies into folders or named components for easier management.
- Use "Join" to create seamless, monolithic parts for structural analysis.
- Use "Cut" to generate features like holes, slots, or complex internal geometries.
- Use "New Body" when designing components intended for assemblies or when parts need individual material properties.
- Combine commands thoughtfully; for example, create a base model with "New Body," then apply "Cut" and "Join" to refine it.

This approach streamlines your workflow and reduces errors.

Comparison of Join, Cut, and New Body in Fusion

360

Feature	Purpose	Typical Use Cases	Effects on Bodies	Best for
Join	Merges multiple bodies into one	Creating continuous parts, welding components	Body becomes a single unified shape	Structural, aesthetic, or seamless designs
Cut	Removes material from a body	Creating holes, slots, internal features	Material is subtracted, leaving a void	Detailing, functional features, modifications

New Body	Creates a separate, independent body	Designing assemblies, multi-part projects, different materials	Bodies remain separate in the model	Multi-component assemblies, independent parts
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Conclusion

Understanding the difference between Join, Cut, and New Body in Fusion 360 is essential for efficient and precise modeling. Whether merging parts seamlessly, subtracting material to create features, or maintaining separate components in a multi-body design, mastering these commands unlocks the full potential of your CAD workflow. Use them thoughtfully to improve your design quality, streamline your processes, and produce accurate, manufacturable models.

FAQ

1. What is the main difference between Join and New Body in Fusion 360?

Ans : Join merges multiple bodies into one, creating a seamless part, while New Body keeps objects separate and independent within the same design.

2. When should I use the Cut operation instead of other features?

Ans : Use Cut when you want to subtract material from an existing body, such as making holes or internal slots.

3. Can I convert one body into another in Fusion 360?

Ans : Yes, using the Combine command with the Join operation, you can fuse bodies into a single shape or convert them into a new body.

4. How does the "New Body" option affect my model's file size?

Ans : Creating multiple bodies increases file size slightly and allows for easier editing of individual components.

5. Is it possible to undo a Join or Cut operation?

Ans : Yes, Fusion 360 supports undo actions, so you can revert changes if needed during your editing session.

6. Can I perform Cut operations using sketches or only bodies?

Ans : You can perform Cut operations with bodies or by extruding sketches set to "Cut" mode for precise internal features.

7. How do these commands relate to assembly modeling?

Ans : "New Body" helps create separate components for assemblies, while "Join" and "Cut" are primarily used for refining individual parts within or across bodies.

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

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2D Sketching

• 3D Modeling



3D Modeling

• Assembly



Assembly

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2D Sketching • 3D Modeling • Assembly Drawings

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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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Customer feedback is critical to our efforts at CADIN360.

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

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EXERCISE-01



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

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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)