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

WHEN TO USE NEW BODY OPTION IN FUSION 360

• LEARN • • APPLY • • GROW •

Introduction

In Fusion 360, the "New Body" option is a fundamental tool for creating complex and separate parts within your design. Knowing when to use the New Body feature can greatly improve your workflow, enable better organization, and facilitate more efficient modifications. Whether you're drafting a multi-part assembly or experimenting with different design variations, understanding the right moments to use New Body is key. This guide will walk you through the when, why, and how of using the New Body option in Fusion 360, providing practical advice and clear steps to help both beginners and experienced designers optimize their CAD modeling process.

What Is the "New Body" Option in Fusion 360?

The "New Body" option allows you to create distinct, separate solid objects within the same Fusion 360 document. When you start a new sketch, extrude, or other solid modeling operations, Fusion 360 offers a toggle for "New Body" versus "Join." Selecting "New Body" keeps the resulting geometry independent of other bodies, rather than combining or cutting into an existing one.

Why Use Separate Bodies Instead of Merging?

Creating multiple bodies within a single design offers several advantages:

- Easier to modify individual parts later
- Facilitates independent analysis or simulation
- Supports assemblies with multiple components
- Allows for easier exporting or manufacturing of individual parts

When to Use the "New Body" Option in Fusion 360

Knowing when to strategically use the New Body feature can significantly enhance your design flexibility. Here are the primary scenarios where employing "New Body" is beneficial.

1. Designing Multi-Component Assemblies

If your project involves assembling multiple parts, such as a housing, a lid, and internal brackets, creating each as a separate body from the start simplifies the process. This approach:

- Keeps parts isolated for easier edits
- Enables you to define different material properties
- Simplifies exporting parts individually for manufacturing

2. Creating Independent Features Within a Single Design

For complex models where certain features are meant to be separate parts, use "New Body." Examples include:

- An internal pipe fitting within a housing
- Multiple extrusions that will later be assembled
- Components that need to be manufactured separately

3. Conducting Simulations and Analysis on Individual Parts

In FEA (Finite Element Analysis) or CFD (Computational Fluid Dynamics), analyzing individual bodies separately can give more accurate insights. Creating each as a new body allows:

- Isolated stress analysis
- Separate meshing and boundary conditions
- Easy modification without affecting other parts

4. Preparing for Manufacturing and 3D Printing

When preparing models for manufacturing, especially 3D printing, keeping bodies separate provides:

- Flexibility to print or process components independently
- Easier assignment of different print parameters

- Simplified assembly post-processing

5. Experimenting with Multiple Design Variations

Designers can create different versions of a component as separate bodies within the same file. This setup simplifies comparisons and iterative testing without needing multiple files.

How to Use the "New Body" Option in Fusion 360

Here's a step-by-step process on effectively using "New Body" in your workflow.

1. Start Creating a New Part

- Begin by selecting the appropriate workspace, such as "Solid" for full modeling.
- Create your initial sketch on a suitable plane.

2. Initiate Sketch and Create Geometry

- Draw your base geometry—circles, rectangles, or complex profiles.
- Finish the sketch once your shape is ready.

3. Use the Extrude Tool with the "New Body" Option

- Select the extrude command.
- In the extrusion dialog box, locate the "Operation" section.
- Choose "New Body" from the drop-down menu.
- Specify the extrusion distance and direction.
- Confirm the operation.

4. Repeat for Additional Bodies

- For each new part or feature, create new sketches.

- Make sure to select "New Body" each time you extrude or perform a feature that results in a new separate component.

5. Managing Multiple Bodies

- Use the "Browser" to see all active bodies.
- Rename bodies for clarity (e.g., "Housing," "Lid," "Bracket").
- Use the "Combine" tool if needed to join or cut bodies later.

6. Exporting and Using Bodies

- When ready to manufacture, you can export each body individually (e.g., STL or STEP files).
- In the "File" > "Export" dialog, select the specific body.

Common Mistakes and How to Avoid Them

Even experienced users can sometimes overlook best practices. Here are common mistakes when using the "New Body" feature and how to avoid them.

1. Forgetting to Select "New Body" During Extrusion

Mistake: All features default to joining, resulting in a single complex body.

Solution: Always double-check the operation setting; select "New Body" before confirming extrusions intended to create separate components.

2. Overusing "New Body" When Not Necessary

Mistake: Creating many tiny or unnecessary separate bodies makes management difficult.

Solution: Use "New Body" only when appropriate. For simple parts, merging may be more practical.

3. Not Naming Bodies

Mistake: Bodies are left unnamed, causing confusion.

Solution: Regularly rename bodies for better organization, especially in complex designs.

4. Neglecting to Use the "Combine" Tool

Mistake: Trying to modify multiple bodies without combining or subtracting.

Solution: Use the "Combine" features to join, cut, or intersect bodies as needed.

Best Practices for Using New Bodies in Fusion 360

To maximize efficiency and organization:

- Plan your design with multiple bodies in mind from the start.
- Use the "Rename" feature to label each body clearly.
- Group related bodies into components for assembly.
- Leverage the "Scene" workspace to display or hide bodies during review.
- When exporting, save each body as a separate file if needed for manufacturing.

Comparing "New Body" With "Join" and "Cut"

Operation	Effect	Best Use Cases

New Body	Keeps geometry separate and independent	Multi-part assemblies, separate features
Join	Combines geometries into one solid	Merging parts into a single object
Cut (Split)	Removes material from existing bodies	Creating holes or subtractive features

Using the correct operation depends on your design intent. Always choose "New Body" when the goal is to keep parts independent.

Conclusion

Knowing when to use the "New Body" option in Fusion 360 is essential for efficient, organized, and flexible 3D modeling. Whether designing multi-part assemblies, preparing for manufacturing, or conducting separate analyses, creating bodies at the right moments simplifies modifications and updates. By understanding the appropriate scenarios, following step-by-step procedures, and avoiding common pitfalls, you can streamline your workflow and achieve better results.

Incorporate these practices into your design process to optimize your Fusion 360 experience and create professional, precise models.

FAQ

1. When should I use "New Body" instead of "Join" in Fusion 360?

Ans: Use "New Body" when you want to keep parts separate for easier editing, analysis, or manufacturing, instead of merging them into a single solid.

2. Can I convert a "New Body" into a component later?

Ans: Yes, you can convert bodies into components by right-clicking the body and selecting "Create Component" or by splitting the body into separate components.

3. How do I combine multiple bodies into one in Fusion 360?

Ans: Use the "Combine" tool with the "Join" operation to fuse multiple bodies into a single solid.

4. Does creating multiple bodies affect the file size or performance?

Ans: Yes, more bodies can slightly increase file complexity and processing time, but it generally doesn't impact performance significantly unless the model is very detailed.

5. Is it possible to change a body from "New Body" to "Join" after creation?

Ans: No, it's not directly possible; you need to use the "Combine" tool afterward to merge bodies if required.

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.

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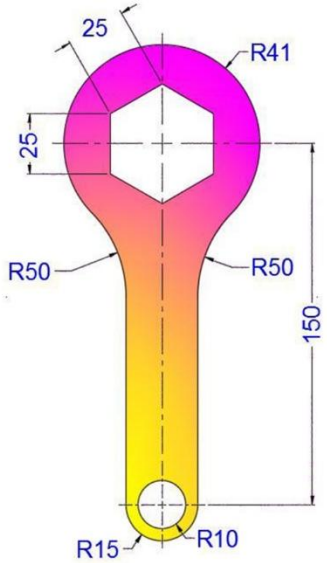
- 3D Modeling
- Assembly



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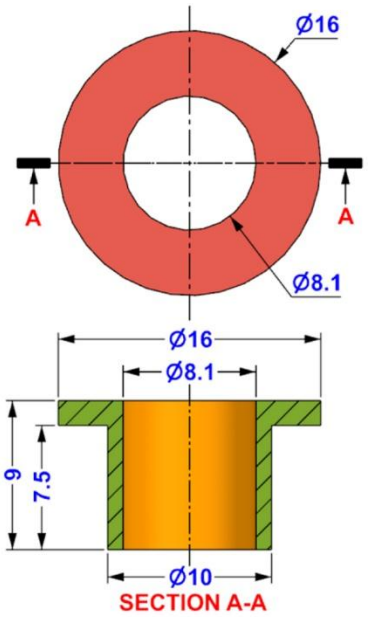
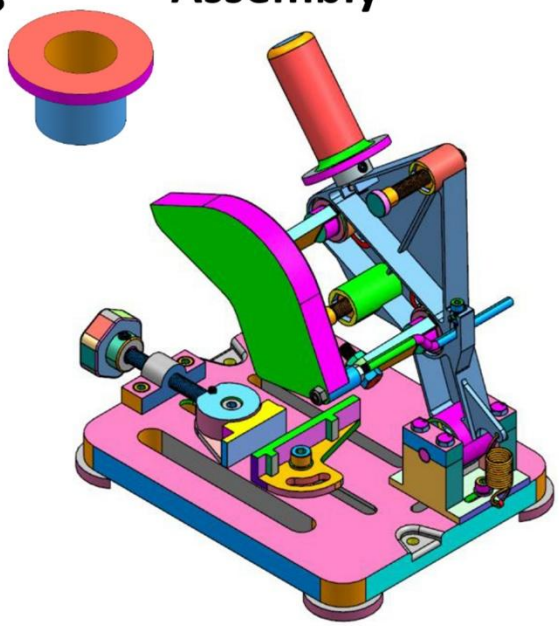


3D Modeling



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:

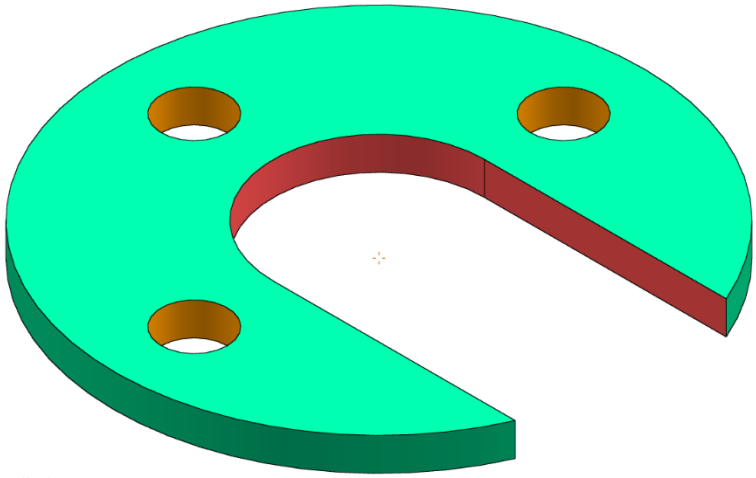
- Complete the exercises in order, or jump to any skill level you prefer.
- All dimensions are in millimeters.
- Where dimensions are missing, apply logic or practice estimation.
- This book is ideal for both students and professionals preparing for industry design work.

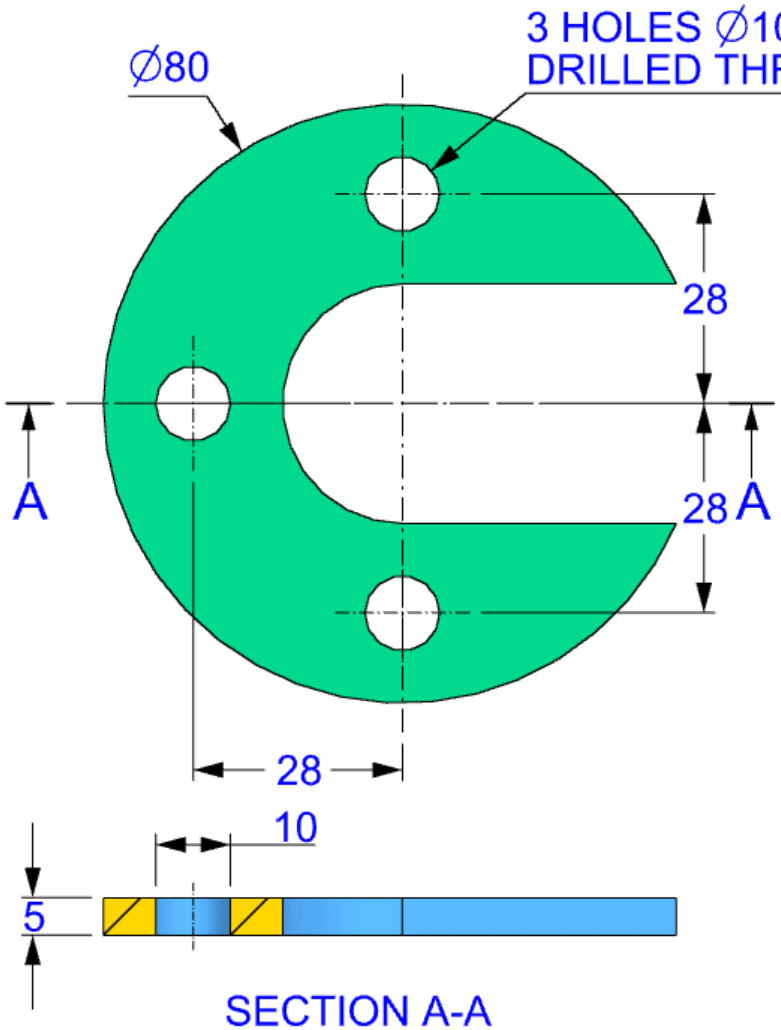
Note:

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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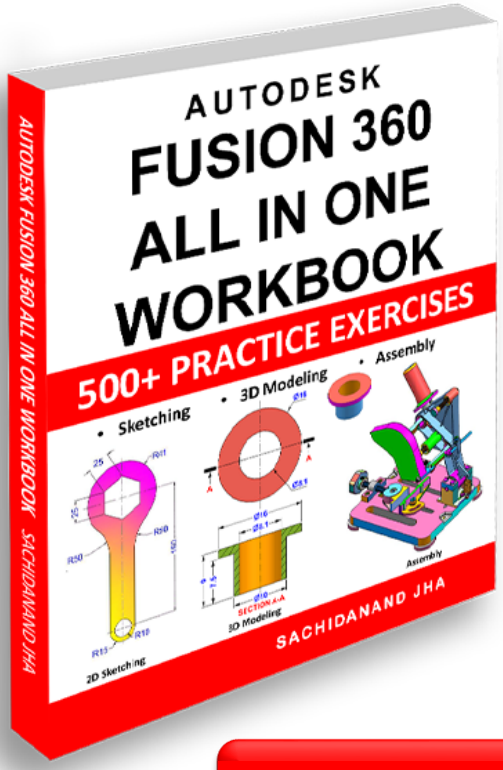

What's Included in the FUSION 360 ALL IN ONE WORKBOOK?

- ✓ 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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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 3rd Angle projection
- All dimensions are in mm(metric system)