AUTODESK FUSION 360

2026

BLOG

WHY EXTRUSION JOINS AUTOMATICALLY IN FUSION 360

• LEARN •   • APPLY •   • GROW •

Introduction

In Fusion 360, a common question for new users is, "Why do extrusion joins automatically?" Understanding this behavior is crucial to mastering the software's capabilities and avoiding common design pitfalls. Fusion 360's automatic joining during extrusion can seem confusing at first, especially for those unfamiliar with its default settings and logical behaviors. This guide dives into why this happens, how to control it, and best practices to optimize your workflow.

Understanding Fusion 360's Extrusion Behavior

Fusion 360 is designed to streamline your modeling process, making it intuitive for both beginners and advanced users. One of its key features is the automatic joining of extrusions, which can surprise those expecting separate bodies by default.

What is extrusion in Fusion 360?

Extrusion is a fundamental operation that extends a 2D sketch into a 3D shape. It allows users to create complex geometry from simple sketches, such as rectangles, circles, or custom shapes.

Why does Fusion 360 join extrusions automatically?

Fusion 360 joins extrusions automatically based on its default settings aiming to simplify the modeling process. When you extrude multiple profiles, the program intelligently determines whether to create separate bodies or to join them into a single, unified body.

This default behavior minimizes errors and saves time, especially when creating parts that are intended to be solid and connected. However, understanding when and why it joins is critical for precise design control.

How Fusion 360 Determines Whether to Join or Keep Separate

Fusion 360's default extrusion command is designed to combine multiple profiles into a single body unless instructed otherwise. Several factors influence this:

1. **Default Operation Setting: The 'Join' option is selected by default during extrusion.**

2. **Selection of Multiple Profiles: Extruding multiple sketches in a single command often results in automatically joining bodies.**

3. **Position of Profiles: Profiles that are adjacent or overlapping tend to be joined by default.**

4. **Type of Extrude Operation: The operation type (Join, Cut, New Body) determines whether bodies are combined.**

The Default 'Join' Operation

When you initiate an extrusion, Fusion 360 defaults to 'Join' unless changed manually. This ensures rapid creation of interconnected parts but requires user awareness to avoid unintended merges.

Step-by-Step Guide: Why Extrusion Joins Automatically and How to Control It

Understanding how and why extrusion joins automatically allows users to control their modeling process effectively.

1. Initiate an Extrude Command

Start by selecting a sketch with one or more profiles.

2. Observe the Default Operation

Notice that in the Extrude dialog box, the 'Operation' option is set to 'Join.'

3. Adjust the Operation Type

- To prevent automatic joining:
- Select the dropdown menu in the Extrude dialog.
- Choose between:

- **New Body**: Creates a separate, standalone body for each extrusion.

- **Cut**: Removes material from an existing body.

- **Join** (default): Merges the new extrusion with existing bodies.

4. Confirm your selection

Click 'OK' to complete the extrusion process based on your chosen operation.

5. Multiple profiles in a single sketch

If your sketch contains multiple closed profiles:

- The default operation joins them all into a single body.

- To keep profiles separate, select 'New Body' for each extrusion.

Practical Examples and Applications

Understanding when and why extrusion joins automatically is beneficial in various scenarios:

Example 1: Designing a Solid Cube with Multiple Holes

- Sketch multiple circles for holes.

- Extrude the entire sketch as 'Cut' to create holes.

- The extrusions of the holes naturally don't join with the main body but cut into it.

Example 2: CreatingComplex Assemblies with Multiple Parts

- Use 'New Body' for each part to keep components separate.

- For welding or merging parts later, use the 'Join' operation at the appropriate stage.

Example 3: Staged Assembly Designs

- First, extrude different parts with 'New Body.'
- Then, assemble and join parts as needed during the modeling process to maintain flexibility.

Common Mistakes and How to Avoid Them

Identifying common pitfalls can save time and frustration.

1. Forgetting to change the operation from 'Join' to 'New Body'

- Always check the operation setting before confirming the extrusion.

2. Extruding multiple profiles unintentionally as one body

- Use separate extrusion commands for each profile or select 'New Body' explicitly.

3. Overlooking overlapping profiles

- Overlapping profiles tend to be joined automatically; separate or offset them if you need distinct bodies.

Best Practices for Managing Extrusions in Fusion 360

- **Always verify the operation type before extruding.**
- **Use 'New Body' for distinct components that need to remain separate.**
- **Utilize selection filters to isolate specific profiles for different operations.**
- **Leverage the 'Create Components' feature for complex assemblies.**
- **Regularly check your bodies and components to ensure they meet design intent.**

Comparing Fusion 360 Extrusion Operations

Operation Type	Default Behavior	Use Case	Advantage
Join	Yes	Creating a solid part from multiple profiles	Simplifies design process
New Body	No (except when selected)	Multiple independent parts within one design	Maintains component separation

Cut	No	Removing material from existing bodies	Precise material removal

Conclusion

Understanding why extrusion joins automatically in Fusion 360 is key to controlling your design process. By recognizing the default behaviors and knowing how to adjust operation settings, you can ensure your models develop exactly as intended. Whether creating interconnected parts or separate components, mastering this aspect of Fusion 360 enables more efficient and accurate modeling.

FAQ

1. Why does Fusion 360 join extrusions by default?

Ans : Fusion 360 defaults to 'Join' operation to combine multiple profiles into a single body, streamlining the modeling process.

2. How can I create separate bodies during extrusion?

Ans : Change the operation from 'Join' to 'New Body' in the extrude dialog box.

3. Can I convert a joined body into separate bodies later?

Ans : Yes, using the 'Split Body' or 'Move/Copy' commands, you can separate parts after extrusion.

4. What happens if I extrude overlapping profiles?

Ans : Overlapping profiles tend to be merged into a single body when extruded with default settings.

5. How do I prevent multiple sketches from automatically joining?

Ans : Use separate extrude commands for each sketch profile and select 'New Body' to keep them separate.

6. Is it possible to change the default extrusion operation globally?

Ans : No, Fusion 360 defaults to 'Join,' but you can set your preferred operation each time in the dialog box.

7. How do I switch between join and new body during an active extrusion?

Ans : Simply open the extrude dialog box during the operation and select your desired operation type before confirming.

By understanding and controlling Fusion 360's extrusion joins, you can optimize your workflows, avoid unintended merges, and produce more precise models tailored to your design objectives.

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.

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

- Sketching
- 3D Modeling
- Assembly

Ø16

25
R41
25
R50
R50
150
R15
R10

2D Sketching

Ø16
A   A
Ø8.1

Ø16
Ø8.1
9
7.5
Ø10
SECTION A-A

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,

Thank you for choosing the AUTODESK FUSION 360 ALL IN ONE WORKBOOK. This book is part of the CADIN360° learning series, created to help engineers, students, and professionals master Fusion 360 through structured and practical exercises.

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

If you think you've found a technical error in this book, please visit https://cadin360.com/contact-us/.
Customer feedback is critical to our efforts at CADIN360.

Best regards,

Sachidanand Jha
Founder & CEO, CADIN360

AUTODESK FUSION 360 ALL IN ONE WORKBOOK

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

3D

EXERCISE-01

3 HOLES ⌀10
DRILLED THROUGH

⌀80

28

28

A

A

28

10

5

SECTION A-A

Get The Complete Practice Sample

You downloaded a single Exercise PDF

The complete practice sample for this software includes multiple exercises and is not available inside this PDF..

What you will receive
- A software-specific complete sample PDF
- Multiple real practice exercises (not a single file)
- Same quality as our professional training material
- Compatible with the latest software version

How to get the complete sample
Click the button below and **enter a** valid email **address**. The **complete sample PDF will be delivered automatically** after the form is submitted.

SEND THE COMPLETE SAMPLE TO MY EMAIL

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
- ✓ Instant Download Link - Sent to Your Email After Payment
- ✓ Lifetime Access to All Files

🙇 Thank You for Learning with Us!

Thank you for choosing the **AutoDesk Fusion 360 ALL IN ONE WORKBOOK**. We hope this book helped you strengthen your Fusion 360 skills through hands-on practice and real-world design challenges.

Your feedback means the world to us!

If you found this book helpful, please take a moment to leave a **review** on the Amazon where you purchased it. Your kind words not only motivate us but also help other learners discover our resources. Scan the QR.

★ A good review goes a long way!

🖼 **Explore More CAD Practice Books**

Looking to continue your learning journey?

We offer similar practice-based books for

over **30 CAD software platforms**, including:

- AutoCAD
- SolidWorks
- FreeCAD
- TinkerCAD
- TurboCAD
- Siemens NX
- CATIA
- Creo
- SketchUp and many more...

Visit our website ☞ **www.cadin360.com** to browse the complete collection.

💬 **Stay Connected**

Have suggestions, feedback, or just want to say hello?

We'd love to hear from you!

✉ Email: cadin360@gmail.com

🌐 Website: www.cadin360.com

🚀 **Keep Practicing. Keep Designing.**

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)