AUTODESK FUSION 360

BLOG

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Learning Tutorials

A Note to Our Readers

2026

HOW TO REMOVE FILLET IN FUSION 360

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Introduction

Removing a fillet in Fusion 360 might seem straightforward, but it can sometimes pose challenges, especially when fillets are integrated into complex models. Whether you need to modify or completely delete a fillet for a design revision or to prepare for further features, understanding the correct method is crucial. In this comprehensive guide, we'll explore **how to remove fillet in Fusion 360** with step-by-step instructions, practical tips, and common pitfalls to avoid. By mastering these techniques, you'll improve your modeling efficiency and ensure clean, precise designs.

Understanding Fillet in Fusion 360

Before diving into removal methods, it's important to understand what a fillet is in Fusion 360. A fillet is a rounded transition between two surfaces or edges, often used for aesthetic purposes, stress reduction, or functional requirements like mounting or assembly.

Key points:

- Fillets are created by selecting edges or faces.
- They can be applied to internal or external edges.
- Once added, fillets are part of the feature history, affecting how they can be modified or removed.

How to Remove Fillet in Fusion 360

Removing a fillet in Fusion 360 can be achieved through different methods depending on your design intent and the model's complexity.

1. Using the "Edit Fillet" Feature

This is the most straightforward approach when the fillet is an independent feature.

- Ideal for models where the fillet was created as a separate feature from the base geometry.

Step-by-step instructions:

1. Open the Timeline

- Locate the **Timeline Bar** at the bottom of your Fusion 360 workspace. Here, all recent features, including fillets, are listed.

2. Find the Fillet Feature

- Scroll through the timeline to identify the **Fillet** feature. It appears as an icon usually labeled "Fillet" or similar.

3. Edit or Delete the Fillet

- To **edit**: double-click the fillet icon. This opens options where you can change the radius to zero or adjust its parameters.

- To **remove**: right-click the fillet icon and select **Delete**. This instantly removes the fillet, restoring the original edges.

4. Confirm Changes

- After deleting or editing, Fusion 360 updates the model in real-time. Review the geometry to ensure the fillet is removed as desired.

2. Editing Fillet Radius to Zero

Sometimes, simply setting the fillet radius to zero effectively removes it.

How to do this:

- Double-click the fillet feature in the timeline.

- In the dialog box, set the **Radius** to **0 mm**.

- Hit **OK** and observe that the fillet disappears.

Note: This method works best for simple fillets and when the feature was created with a specific radius.

3. Deleting the Fillet Using Direct Modeling (For Complex Models)

If the fillet is a result of a complex geometry or a baked feature, you might need to use **direct modeling** techniques.

Practical steps:

1. Enter the "Modify" Panel

- Go to the **Modify** drop-down menu in the toolbar.

2. Use the "Delete Face" or "Delete Edge" Tools

- Select **Delete Face** to remove faces associated with the fillet or **Delete Edge** to remove the specific filleted edge.
- Confirm by clicking on the relevant face or edge in the model.

3. Heal the Geometry

- Fusion 360 will attempt to heal the model automatically. If gaps or issues arise, use tools like **Replace Face** or **Patch** to fix.

Note: These methods are more complex and less predictable; they are best suited for experienced users or complex repairs.

Practical Examples and Scenarios

Example 1: Removing a Simple External Fillet

Suppose you created an external rounded edge on a cube for ergonomic purposes. To remove it:

- Locate the "Fillet" feature in the timeline.

- Right-click and delete, or edit radius to zero.

Example 2: Dealing with Internal Fillet

For interior fillets in a hollow part:

- Find the fillet feature.

- Use the same delete or zero-radius approach.

- Be cautious: deleting internal features can sometimes affect internal geometry or other features.

Example 3: Removing Multiple Fillets

When multiple fillets exist:

- Repeat the delete or edit process for each.

- To efficiently delete several features, select the first, then hold **Shift** and click additional features to select multiple fillets before deleting.

Common Mistakes and How to Avoid Them

- **Deleting the wrong feature**: Confirm the feature in the timeline before deletion.

- **Forgetting to rebuild after deletion**: Fusion 360 often updates automatically, but ensure your model regenerates cleanly.

- **Removing essential features unintentionally**: Check dependencies; some fillets may link with other features.

- **Editing instead of deleting**: Sometimes setting radius to zero doesn't give the desired result if the fillet was baked or part of a complex feature.

Tips and Best Practices for Removing Fillet in Fusion 360

- Always **save a backup** before making extensive deletions or modifications.
- Use **parametric approach**: editing parameters gives more control than outright deletion.
- When working with complex models, **isolate features** using component or body selection to prevent accidental modifications elsewhere.
- If a fillet cannot be removed through traditional means, consider **recreating the geometry** or using direct modeling techniques.

Comparison: Editing vs. Deleting Fillet in Fusion 360

Method	Pros	Cons	Best Use Case
Edit Fillet (Radius Zero)	Quick, preserves feature history	Might leave unused features at zero	Simple, isolated fillet removal

Delete the Fillet Feature	Clearest, removes completely	Loss of parameter history	When the fillet is no longer needed
Direct Modeling	Handles complex geometry	Riskier, requires experience	For baked or complex models

Conclusion

Removing a fillet in Fusion 360 is a common task that can be accomplished efficiently using the right approach. Whether editing the feature to zero radius, deleting the feature directly from the timeline, or applying more advanced techniques for complex geometries, understanding these methods will give you greater control over your design process. Always approach fillet removal with caution, especially in intricate models, and remember to save backups before making significant changes. With practice, you'll be able to swiftly modify your designs to meet your evolving needs.

FAQ

1. How do I remove a fillet from a specific edge in Fusion 360?

Ans: Use the timeline, double-click the fillet feature, and either delete it or set the radius to zero.

2. Can I remove multiple fillets at once in Fusion 360?

Ans: Yes, select multiple fillet features in the timeline by holding Shift, then delete or edit them collectively.

3. What is the best way to delete a fillet if it was baked into the geometry?

Ans: Use direct modeling tools like "Delete Face" or "Delete Edge," but be cautious of resulting geometry issues.

4. How can I prevent accidental deletion of important features?

Ans: Always carefully select features in the timeline and consider creating different versions or backups before making significant edits.

5. Is it possible to restore a deleted fillet in Fusion 360?

Ans: If you haven't saved or exited, use Ctrl+Z to undo. Otherwise, you'll need to recreate the filament manually.

6. Does deleting a fillet affect parametric constraints?

Ans: Yes, deleting a feature removes its parameters, so rebuild or adjust related features accordingly.

7. Can I use the "Patch" tool to remove a fillet?

Ans: The "Patch" tool can cover gaps but isn't typically used to remove fillets; it's more for repairing geometry.

By following these step-by-step methods, practical tips, and understanding common pitfalls, you'll be well-equipped to master how to remove fillets in Fusion 360 and keep your models clean and precise.

About CADIN360

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

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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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.

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Best regards,

Sachidanand Jha
Founder & CEO, CADIN360

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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

Ø80

3 HOLES Ø10
DRILLED THROUGH

28

28 A

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5

SECTION A-A

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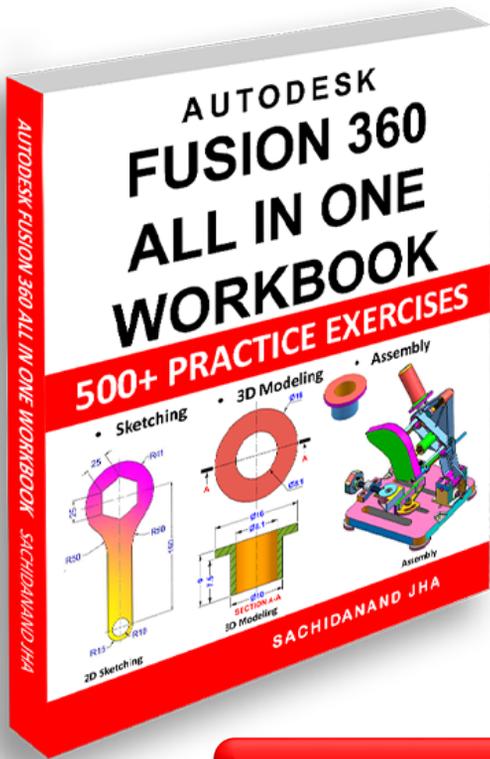
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🚀 **Keep Practicing. Keep Designing.**

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)