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

HOW TO APPLY FILLET TO MULTIPLE EDGES IN FUSION 360

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Introduction

Applying fillet to multiple edges in Fusion 360 is a common task for refining your 3D models. Whether you're designing ergonomic products or smoothening complex geometries, knowing how to efficiently apply fillets to multiple edges saves time and improves your design quality. While Fusion 360 offers straightforward options for single-edge fillets, handling multiple edges requires understanding specific tools and techniques to work efficiently. This guide provides step-by-step instructions, practical tips, and common pitfalls to help you master applying fillets to multiple edges in Fusion 360.

Understanding Fillets in Fusion 360

Before diving into the process, it’s important to understand what a fillet is. A fillet is a rounded internal or external corner at the intersection of two or more surfaces or edges. In Fusion 360, fillets smooth out sharp edges by creating a rounded transition, adding both aesthetic appeal and functional benefits like reducing stress concentrations.

Applying fillet to multiple edges involves selecting several edges simultaneously or sequentially and ensuring the desired radius is consistent or tailored for each. Knowing how to handle these options efficiently is crucial for complex models.

How to Apply Fillet to Multiple Edges in Fusion 360: Step-by-Step Guide

Applying fillet to multiple edges can be done using either the Fillet tool or the Continuous Fillet option. Here are the detailed steps:

1. Prepare Your Model

- Ensure your model is fully modeled and surfaces are clean.
- Fix any geometry issues that might interfere with edge selection, such as open gaps or overlapping faces.

2. Select the Fillet Tool

- Navigate to the **Solid** tab on the toolbar.
- Click on **Fillet** from the **Modify** dropdown menu.

Or use the shortcut by pressing **F**.

3. Select Multiple Edges

- In the canvas, hover over the edges you want to fillet.
- Click on each edge while holding **Ctrl** (Windows) or **Cmd** (Mac) to select multiple edges.

Alternatively:

- Drag a selection box around multiple edges.
- Use the **Selection Filters** to pick only edges.

Tip: Fusion 360 allows for multi-edge selection in the graphics window, but it can become tedious if edges are not close. Use selection filters to improve accuracy.

4. Adjust the Fillet Radius

- Once the edges are selected, move the **Radius** slider in the dialog box.
- Input a specific value for the fillet radius.
- To apply different radii per edge, you need to select edges one at a time and adjust individually, but this isn't possible directly in the multi-edge selection.

5. Use the Continuous Fillet Option (for smooth transitions across multiple edges)

- In the fillet options, select **Continuity** (G0, G1, or G2) for different smoothness levels.
- The **Continuous Fillet** creates seamless, flowing transitions.

6. Confirm and Finalize

- Check your model preview.
- Click **OK** to apply the fillet.

7. Edit Fillets if Needed

- If changes are necessary, double-click the fillet feature in the **Timeline**.
- Adjust the radius or selection as required.

Practical Example: Filleting Multiple Edges on a Mechanical Part

Suppose you are designing a bracket with several sharp edges that need rounding for safety and aesthetics:

- After modeling the bracket, select all sharp external edges.
- Use the multi-selection method to pick edges simultaneously.
- Set a consistent fillet radius, say 2 mm.
- For a more natural transition, choose **G1 continuity**.
- Apply and review the result.
- If some edges need different radii, select them individually and adjust the radius before confirming.

Common Mistakes When Applying Fillet to Multiple Edges

Understanding what can go wrong helps avoid pitfalls:

- **Selecting incompatible edges**: Mistakenly selecting edges that aren't adjacent or don't meet at sharp corners.
- **Choosing an inappropriate radius**: Larger radii can distort the geometry or create overlap.
- **Applying fillet after complex modeling**: Sometimes it's better to consider fillet placement during initial design.
- **Not checking the preview**: Always preview the fillet before confirming to avoid model distortions.

Pro Tips for Efficient Filleting in Fusion 360

- **Use selection filters**: Quickly isolate edges for precise selection.
- **Leverage the push-pull tool**: For complex geometries, push or pull faces before applying fillet.
- **Create construction geometry**: Use construction lines or planes to organize edge selections.
- **Apply fillets progressively**: For complex models, apply smaller radius fillets in steps.
- **Utilize the Sheet Metal environment**: For sheet metal designs, specific fillet tools may give better control.

Comparing Fillet and Chamfer in Fusion 360

Feature	Fillet	Chamfer

Purpose	Creates rounded edges	Creates beveled edges
Use case	Smooth, rounded transitions	Sharp or angled transitions
Impact on geometry	Adds curvature	Adds angled surface
Best for	Safety, aesthetics, stress distribution	Mechanical fits, aesthetics

Understanding the differences helps in choosing the right option for your design needs.

Conclusion

Learning how to apply fillet to multiple edges in Fusion 360 enables designers and engineers to create smoother, more refined models with precision and efficiency. Using selection techniques,

adjusting radii, and understanding the nuances between different fillet types can significantly improve your workflow. Remember to leverage Pro tips to speed up your process, and always verify your results through the preview before finalizing.

Mastering multi-edge filleting ensures your models not only look professional but also function effectively in real-world applications, from consumer products to precision machinery.

FAQ

1. How do I select multiple edges for a fillet in Fusion 360?

Ans: Hold down **Ctrl** (Windows) or **Cmd** (Mac) and click on each edge to select multiple edges simultaneously.

2. Can I apply different fillet radii to each edge in a single step?

Ans: No, Fusion 360 applies a uniform radius to all selected edges in a multi-edge fillet; individual radii require selecting edges separately.

3. What is the best way to create smooth transitions across multiple edges?

Ans: Use the **Continuous Fillet** option with G1 or G2 continuity for seamless, flowing surfaces.

4. Why is my fillet failing on certain edges?

Ans: The geometry may be incompatible, or the fillet radius is too large, causing overlaps or distortions.

5. How do I edit a fillet after applying it?

Ans: Double-click the fillet feature in the **Timeline**, then adjust the radius or edge selection as needed.

6. Is there a way to apply multiple fillets with different radii quickly?

Ans: The most efficient method is to apply separate fillet features per radius, as Fusion 360 does not support multiple radii in a single multi-edge fillet.

7. Can I undo a fillet if I’m not satisfied?

Ans: Yes, select the fillet feature in the **Timeline** and delete or modify it as needed.

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

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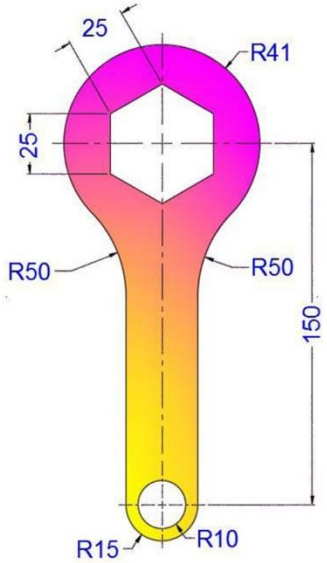
- Assembly



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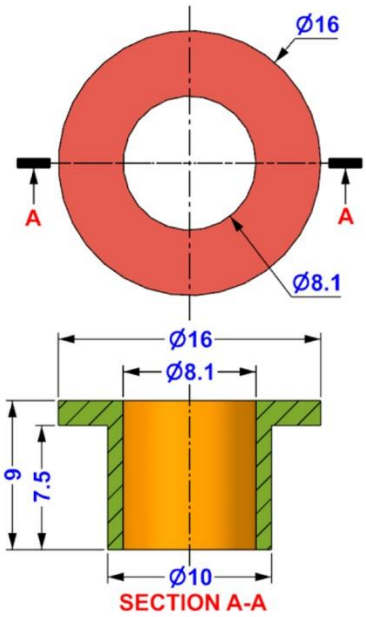

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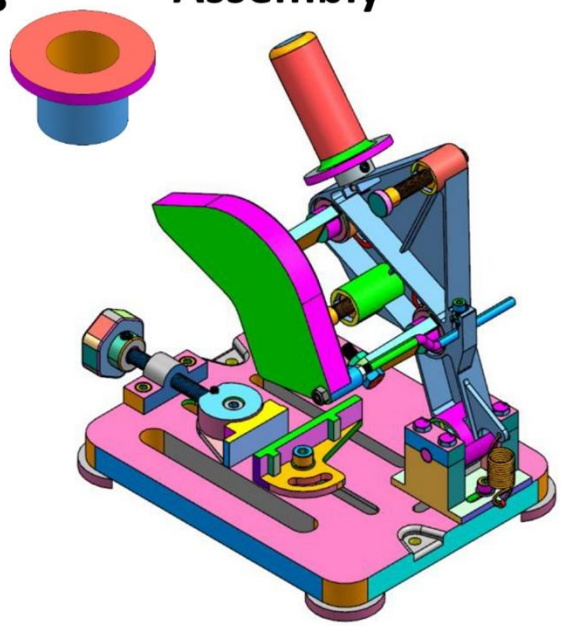

Assembly

SACHIDANAND JHA

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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

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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.

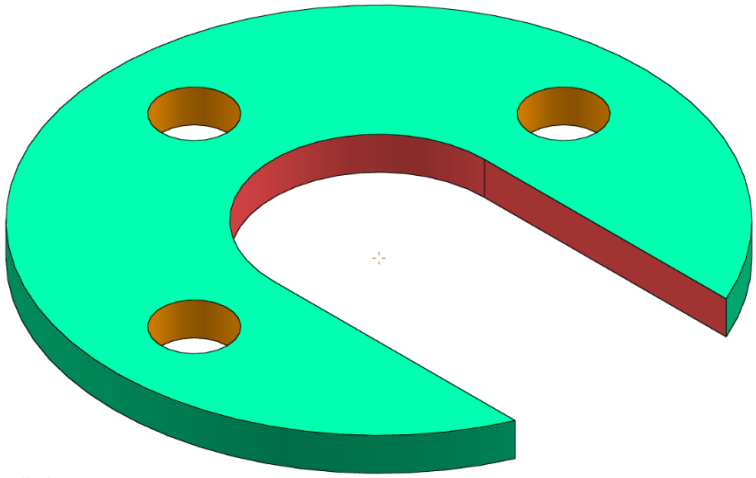
Tips for Best Use:

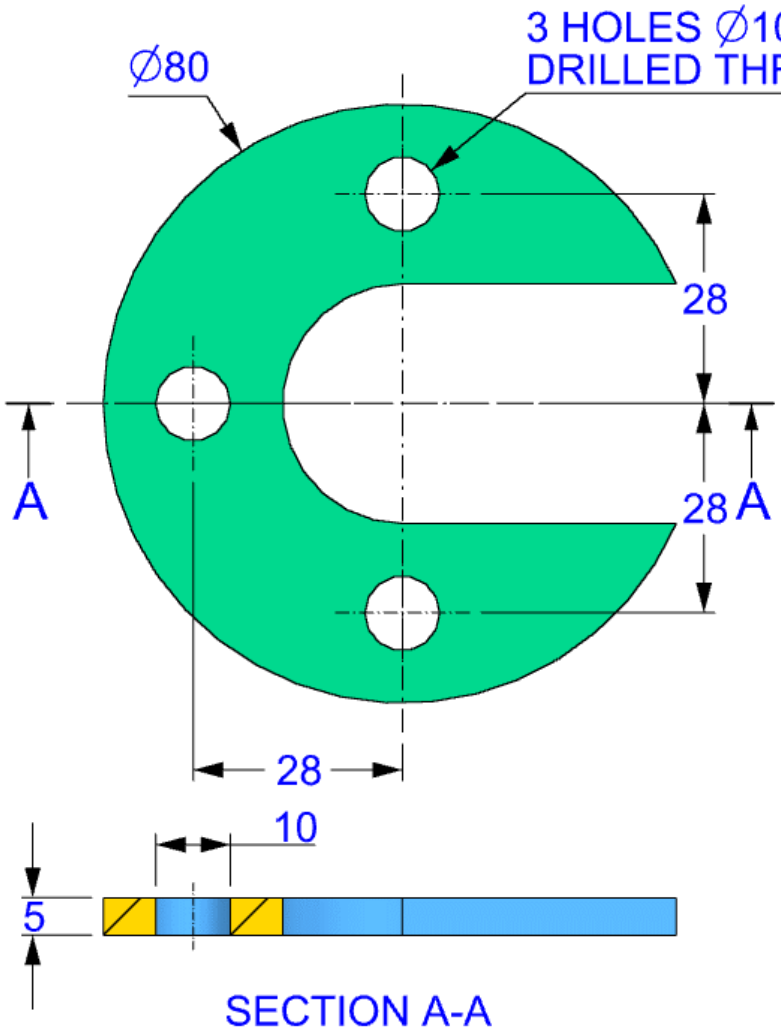
- Complete the exercises in order, or jump to any skill level you prefer.
- All dimensions are in millimeters.
- Where dimensions are missing, apply logic or practice estimation.
- This book is ideal for both students and professionals preparing for industry design work.

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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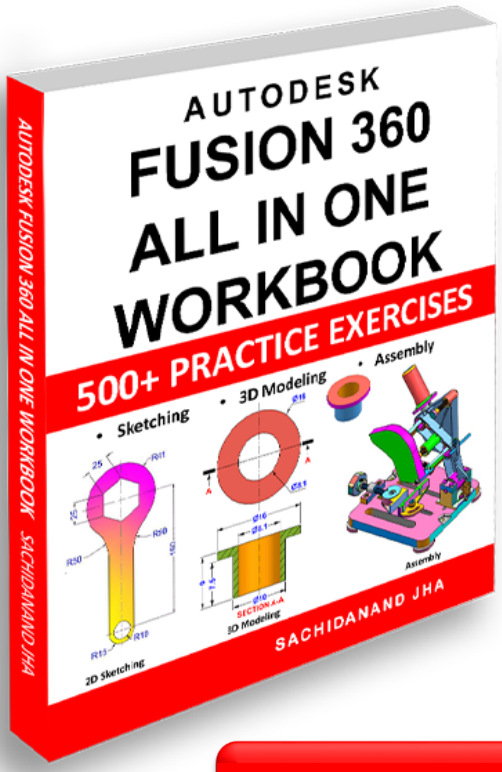

What's Included in the FUSION 360 ALL IN ONE WORKBOOK?

- ✓ 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
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We're honored to be a part of your journey.
Happy Designing!
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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)