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 CHAMFER AND FILLET IN FUSION 360

• LEARN • • APPLY • • GROW •

Introduction

When designing 3D models in Fusion 360, understanding how to refine edges and corners is crucial for both aesthetic appeal and functional performance. Two essential features used to modify edges are chamfers and fillets. **Difference between chamfer and fillet in Fusion 360** is a common question among beginners and experienced designers alike. While both methods smooth out or modify edges, they do so in fundamentally different ways, with distinct applications and outcomes. Mastering these tools enables you to create more precise, manufacturable, and visually appealing parts.

In this comprehensive guide, we delve into the detailed differences between chamfer and fillet in Fusion 360, how to apply each, their practical use cases, and step-by-step instructions. Additionally, we explore real-world examples, common mistakes, and industry best practices to help you make informed decisions in your CAD workflow.

Understanding the Basic Concepts: Chamfer vs. Fillet in Fusion 360

Before diving into step-by-step procedures, it's important to understand what chamfers and fillets are fundamentally.

What is a Chamfer?

A chamfer is a beveled edge that slants or cuts across a corner or edge. It is typically used to remove sharp edges, facilitate assembly, or improve the aesthetic look of a part. Chamfers are created at specific angles and distances, giving a crisp, angled transition between surfaces.

What is a Fillet?

A fillet is a rounded curve applied to the edge or corner of a part. The purpose of a fillet is to smooth out sharp edges, reduce stress concentrations, and improve safety by eliminating sharp corners. Fillets are defined by their radius, creating a smooth, curved transition between surfaces.

How to Create and Apply Chamfer in Fusion 360

Applying a chamfer to your model can be done with a straightforward process. Follow these steps:

1. **Open Your Model**

Launch Fusion 360 and open the design where you want to add a chamfer.

1. **Select the Edges or Corners**

In the Model workspace, click on the edge or corner where you want the chamfer. You can select multiple edges simultaneously.

1. **Activate the Chamfer Tool**

- Navigate to the “Modify” menu.
- Choose “Chamfer” from the dropdown options.

1. **Configure Chamfer Parameters**

- **Distance Distance:** Specifies the length of the chamfer along each edge.
- **Angle or Distance Combo:** Alternatively, you can select a specific angle and distance for precise control.
- **Set Parameters:** Adjust these values as needed for your design.

1. **Preview and Confirm**

Use the preview to visualize the chamfer. If everything looks correct, click OK. If not, tweak the parameters or reselect edges.

Practical Example of a Chamfer

Designing a machine housing with a beveled edge for easier assembly and handling. Adding a chamfer prevents sharp edges that could cause cuts or damage.

How to Create and Apply a Fillet in Fusion 360

Creating a fillet is equally simple but focuses on smooth transitions.

1. **Open Your Model**

Launch Fusion 360 and select your part.

1. **Select the Edges or Corners**

Click on the edge(s) you wish to Bfinish with a fillet.

1. **Activate the Fillet Tool**

- Go to the “Modify” menu.
- Select “Fillet”.

1. **Specify the Radius**

- Enter a radius value, which determines the curvature of the fillet.
- Use the drag handle in the model for visual adjustment if available.

1. **Preview and Apply**

Confirm the shape and curvature visually, then click OK to finalize.

Practical Example of a Fillet

Applying a fillet to the edge of a smartphone case to eliminate sharp corners, making it safer and more comfortable to hold.

Step-by-Step Comparison: Chamfer vs. Fillet

Feature	Chamfer	Fillet
Shape	Straight beveled edge	Rounded curve
Control parameters	Distance, angle	Radius
Use case	Aesthetic, assembly, manufacturing	Safety, stress reduction, smooth finish
Visual outcome	Sharp, angled transition	Smooth, curved transition

Common in industries	Mechanical parts, tools, machinery	Consumer electronics, automotive, aerospace

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Practical Tips for Choosing Between Chamfer and Fillet

- **When to Use a Chamfer**: Choose a chamfer when you want a sharp, angled edge for aesthetic reasons, or when it's necessary for part assembly or machining.
- **When to Use a Fillet**: Use a fillet to reduce stress concentration points, improve safety, or create a smooth transition for aesthetic or ergonomic reasons.
- **Design Considerations**: Think about manufacturing constraints—fillets are often preferred in injection molding and casting, while chamfers are common in machining for easy tooling access.

Common Mistakes and How to Avoid Them

1. **Applying Too Large a Radius/Distance**

- Oversized chamfers or fillets can distort the part or interfere with assembly.

1. **Changing Parameters After Creation**

- Always double-check your parameters before finalizing to avoid rework.

1. **Not considering manufacturability**

- Ensure your chosen edge modifications are feasible with the manufacturing process.

1. **Overusing or misusing these features**

- Use intentionally; too many chamfers or fillets can clutter your design.

Best Practices and Pro Tips

- **Combine both features judiciously** for complex parts, such as using chamfers on mating edges and fillets on stress points.
- **Use visual feedback** during modeling to make real-time adjustments.
- **Apply consistent parameters** across your model to maintain uniformity.
- **Document your features** with comments for easy revisions later.

Conclusion

Understanding the fundamental difference between chamfer and fillet in Fusion 360 is crucial for precise modeling, functional integrity, and aesthetic quality. While both serve to modify edges, their applications and outcomes differ significantly. Chamfers create sharp, beveled edges ideal for assembly and manufacturability, whereas fillets produce smooth, rounded corners that enhance safety, stress distribution, and visual appeal.

By mastering these tools through step-by-step application, practical examples, and adherence to best practices, you can elevate your CAD modelling skills, resulting in better-designed and more manufacturable parts.

FAQ

1. What is the main difference between a chamfer and a fillet in Fusion 360?

Ans: A chamfer creates a straight, beveled edge at an angle, while a fillet produces a rounded, curved edge.

2. When should I use a chamfer instead of a fillet?

Ans: Use a chamfer when an angled edge is needed for aesthetics, assembly, or machining accessibility.

3. How do I create a fillet with a specific radius in Fusion 360?

Ans: Select the edge, activate the "Fillet" tool, then enter the desired radius value in the dialog box.

4. Can I apply both chamfer and fillet to the same model?

Ans: Yes, but use them thoughtfully to maintain clarity, functionality, and manufacturability of the part.

5. How do chamfers and fillets affect manufacturing?

Ans: Chamfers are easier in machining for beveled edges, while fillets are advantageous in casting or molding for smoother transitions.

6. What are common mistakes when applying chamfers or fillets?

Ans: Applying excessively large parameters, overusing the features, and ignoring manufacturability constraints.

7. Can I edit a chamfer or fillet after applying it?

Ans: Yes, both features are parametric and can be easily edited by selecting them in the timeline or feature tree and adjusting parameters.

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.

AUTODESK

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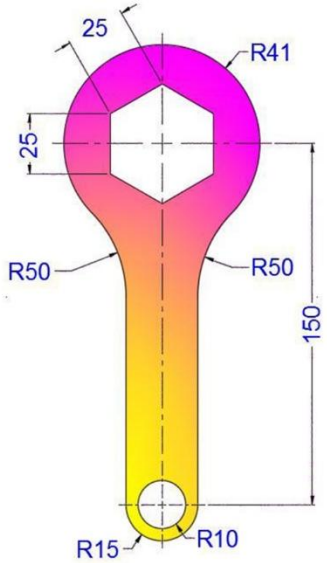
- 3D Modeling
- Assembly



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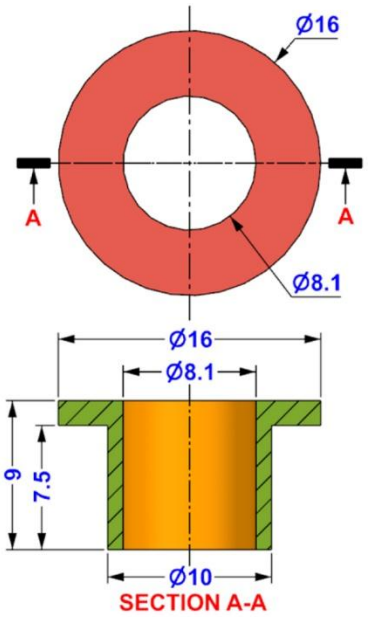
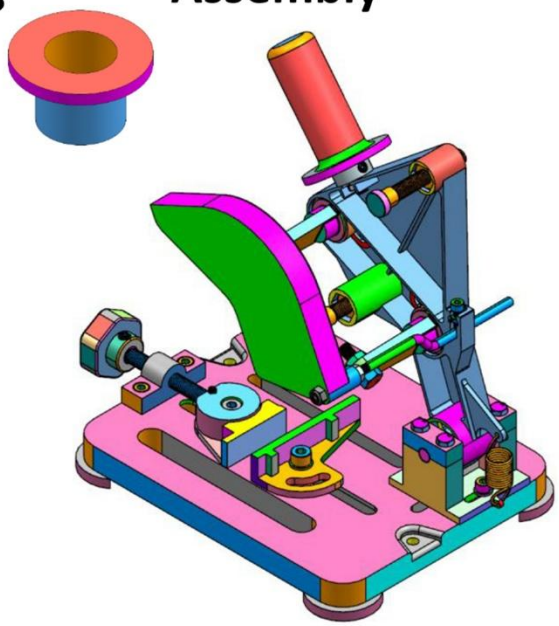


3D Modeling



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:

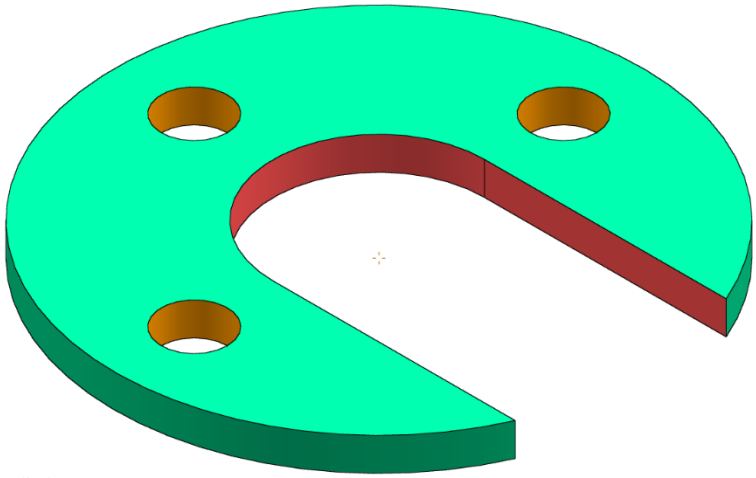
- Complete the exercises in order, or jump to any skill level you prefer.
- All dimensions are in millimeters.
- Where dimensions are missing, apply logic or practice estimation.
- This book is ideal for both students and professionals preparing for industry design work.

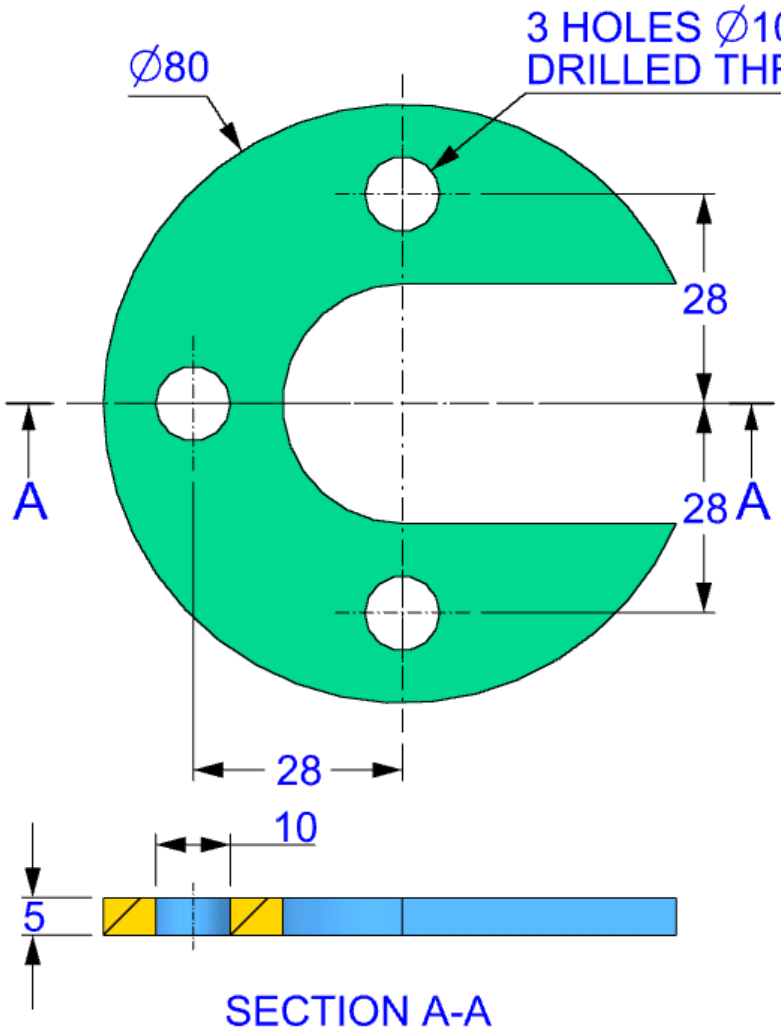
Note:

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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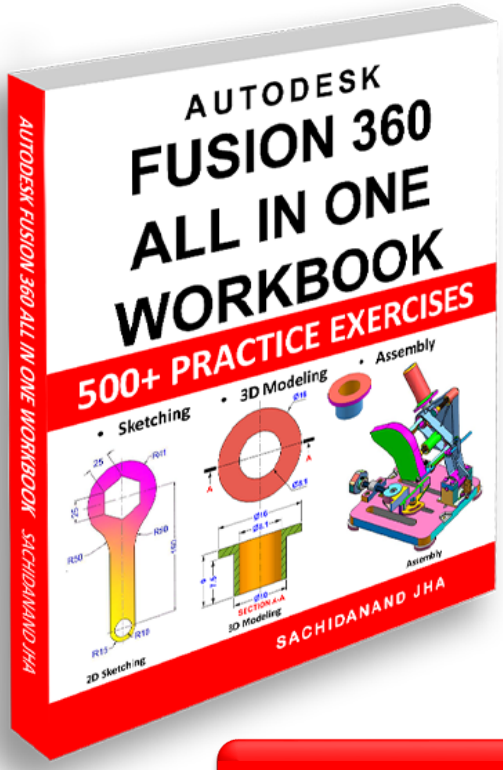

What's Included in the FUSION 360 ALL IN ONE WORKBOOK?

- ✓ 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
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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)