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 SINGLE EDGE IN FUSION 360

• LEARN • • APPLY • • GROW •

Introduction

Applying a fillet to a single edge in Fusion 360 is a common task that enhances the aesthetics and functionality of your 3D models. Whether you are refining a design, softening sharp corners, or preparing parts for 3D printing, mastering the process of applying a fillet to just one edge is an essential skill for CAD users. In this comprehensive guide, we will walk you through the step-by-step process on how to apply a fillet to a single edge in Fusion 360, along with tips, common mistakes, and practical examples. By following these instructions, you'll be able to produce cleaner, professional-looking models that meet your design specifications.

How to Apply a Fillet to a Single Edge in Fusion 360

Applying a fillet to a single edge might seem straightforward, but understanding the nuances will help you achieve precise results. Here's how you can do it efficiently:

1. Prepare Your Model

- Open your existing Fusion 360 project or create a new model.
- Ensure your model is fully modeled and the specific edge you want to fillet is clearly identified.
- Use the browser tree to isolate features if needed for better visibility.

2. Enter the Fillet Tool

- Move to the "Solid" tab in the toolbar.
- Click on the "Fillet" tool. Alternatively, you can access it through the shortcut by pressing **S** and searching for "Fillet."
- The Fillet dialog box will appear, ready for you to select edges.

3. Select the Specific Edge

- In the graphics window, click on the edge you'd like to fillet.

- Ensure you click precisely on the edge to avoid selecting unintended edges.
- If multiple edges are accidentally selected, you can deselect using **Ctrl + click** on the unwanted edge or manually remove the selection from the dialog.

4. Adjust the Fillet Radius

- Move the slider or type in a precise value for the radius in the input box.
- Observe the preview; Fusion 360 dynamically shows the fillet as you adjust.
- Keep in mind that very large radii might cause geometry conflicts, so choose a value appropriate to your design.

5. Confirm the Fillet

- After adjusting the radius to your satisfaction, click "OK" to apply the fillet.
- Fusion 360 will process the change and display the result.

6. Fine-Tune if Necessary

- If the fillet doesn't meet your expectations, undo with **Ctrl + Z**.
- Repeat the process with different radius values or select a different edge if required.
- For more control, you can enter a specific value or use the "Fillet Type" options like "Constant Radius" or "Variable Radius."

Practical Real-World Examples of Applying Single Edge Fillets

Applying a fillet to a single edge is often used in:

- **Automotive part design:** Softening sharp edges for safety and durability.
- **Jewelry modeling:** Creating smooth transitions for aesthetic appeal.

- **Mechanical component design:** Ensuring stress distribution by filleting sharp corners.
- **Product prototypes:** Achieving realistic, manufacturable edges.

Example: Creating a Rounded Corner on a Box

Suppose you have a rectangular box, and you want to apply a fillet to just one vertical edge for aesthetic purposes.

- Select the vertical edge.
- Set a radius of 5 mm.
- Confirm the fillet, and observe the edge transition.
- Finishing this effect enhances the overall look and feel of the product.

Common Mistakes When Applying a Single Edge Fillet

Understanding what to avoid will save you time and improve your workflow:

- **Selecting multiple edges unintentionally:** Always verify your selection before confirming.
- **Choosing an inappropriate radius:** Large radii can cause geometry conflicts or unintended deformation.
- **Applying fillets to complex geometries:** Ensure the edge is suitable for filleting; some edges may have geometry conflicts.
- **Not using the correct edge:** Confirm you're selecting the right edge, especially on models with many curves.

Pro Tips and Best Practices

- Use **zoom and orbit** tools to precisely select edges.
- For complex models, temporarily hide surrounding features for better visibility.

- Utilize the **Press Pull** tool to simulate how the fillet will look on actual parts.
- Explore **Fillet options** like tangent chain to extend fillet continuity across connected edges.
- Always **save a backup** before making extensive modifications.

Fusion 360 Fillet Versus Other CAD Software

Feature	Fusion 360	SolidWorks	Creo
Ease of use	Intuitive, good for beginners	Slightly steeper learning curve, professional tools	Advanced, suitable for complex geometries

Edge selection	Precise, direct edge selection	Similar, with more options for chain selection	Supports complex continuous fillets
Customization	Multiple fillet types, variable radii	Extensive options for fillet types	Advanced control over fillet behavior
Cost	Free for hobbyists / Subscription-based	Paid, with professional licensing	Paid, enterprise solutions

Note: Fusion 360's interface emphasizes simplicity and accessibility, making it ideal for beginners and intermediate users seeking straightforward edge filleting.

Conclusion

Mastering how to apply a fillet to a single edge in Fusion 360 is a foundational skill that enhances your 3D modeling capabilities. By following the step-by-step guide outlined above, you can easily soften sharp edges, improve aesthetic appeal, and prepare your designs for manufacturing. Remember to always select edges carefully, choose appropriate radii, and utilize Fusion 360's dynamic preview to achieve precise results. Whether you're designing functional mechanical parts or aesthetic jewelry, applying a single-edge fillet will elevate your modeling quality and efficiency.

FAQ

1. How do I select a specific edge for filleting in Fusion 360?

Ans: Click directly on the edge in the graphics window, ensuring no other unintended edges are selected.

2. Can I apply a different radius to multiple edges in Fusion 360?

Ans: Yes, after selecting multiple edges, you can set different radii for each in the fillet dialog or use the "Variable Radius" option.

3. What should I do if the fillet causes geometry conflicts or errors?

Ans: Try reducing the radius size, ensuring the edge geometry is clean, or check for overlapping features that might interfere.

4. Is it possible to fillet a curved or rounded edge in Fusion 360?

Ans: Yes, Fusion 360 supports fillets on curved or rounded edges, but complex curvature may require careful selection and additional smoothing.

5. How can I preview the fillet before confirming in Fusion 360?

Ans: Fusion 360 automatically previews the fillet as you adjust the radius; ensure the preview looks correct before clicking "OK."

6. Can I apply a fillet to an internal or hidden edge?

Ans: Yes, but you might need to unhide edges or rotate the view for better selection accuracy; internal edges may require special attention.

7. Does applying a fillet affect the underlying geometry for further edits?

Ans: Yes, applying a fillet modifies the geometry, making subsequent edits to that edge or radius easier within the feature tree.

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

FUSION 360 ALL IN ONE WORKBOOK

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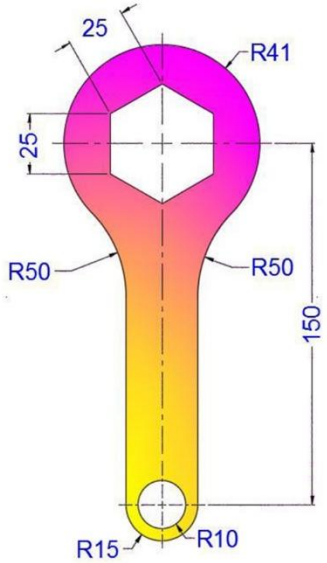
- 3D Modeling
- Assembly



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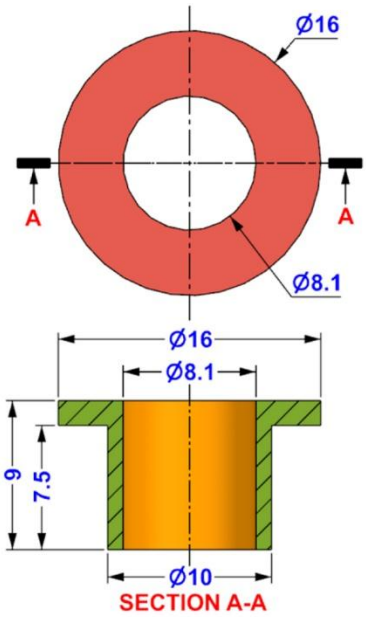

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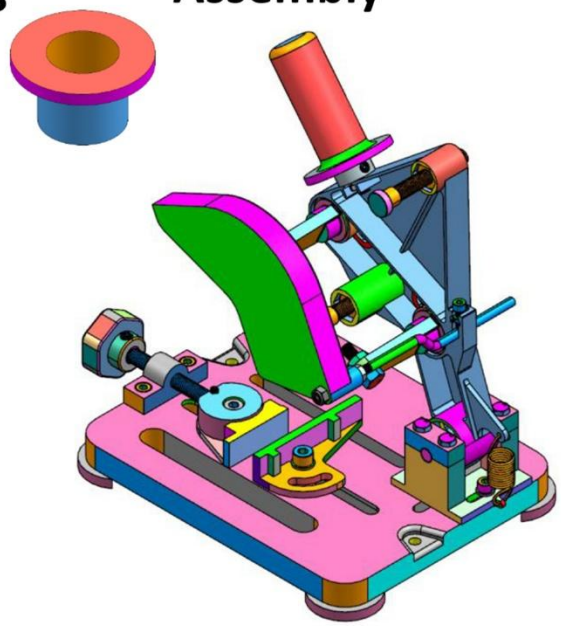

Assembly

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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Customer feedback is critical to our efforts at CADIN360.

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.

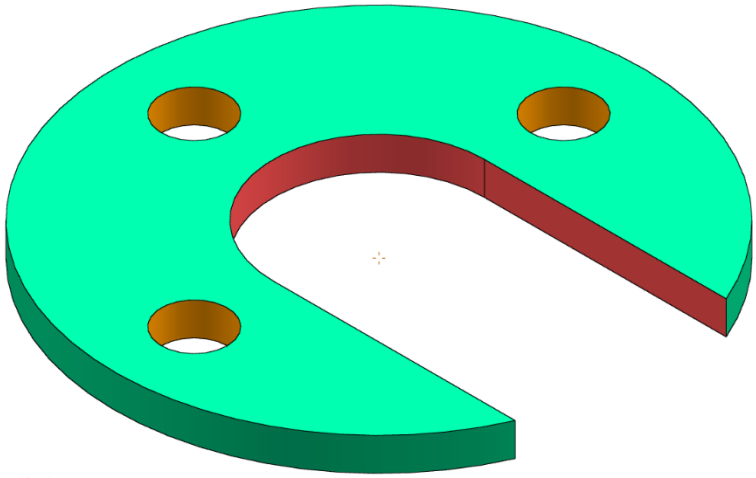
Tips for Best Use:

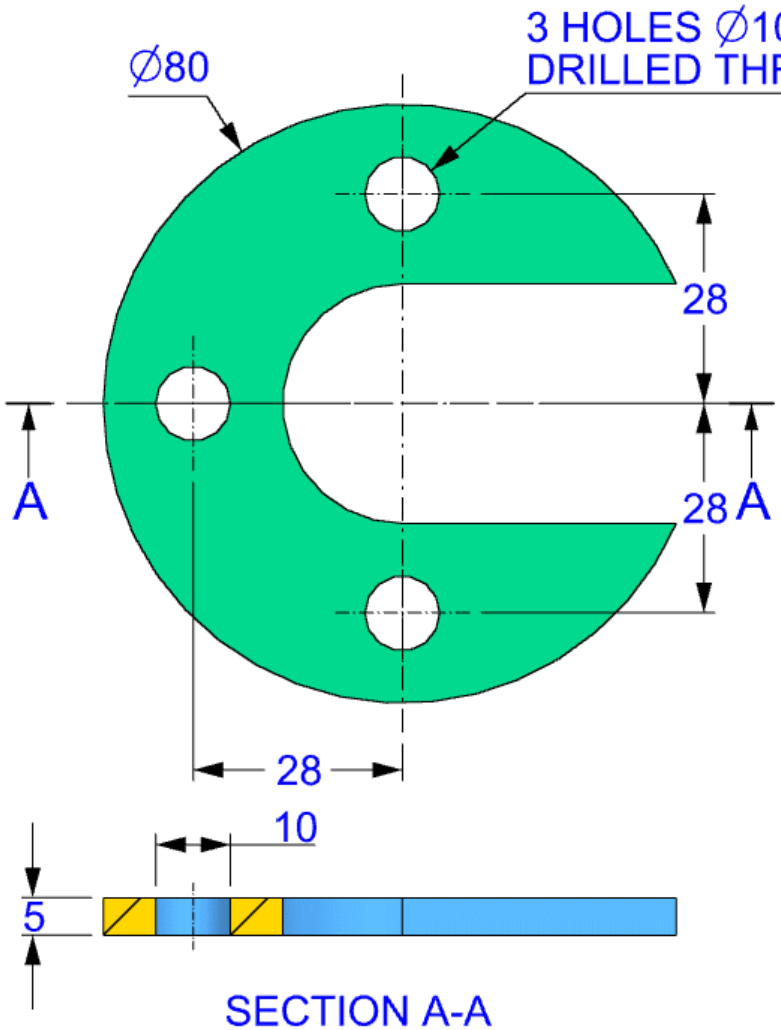
- Complete the exercises in order, or jump to any skill level you prefer.
- All dimensions are in millimeters.
- Where dimensions are missing, apply logic or practice estimation.
- This book is ideal for both students and professionals preparing for industry design work.

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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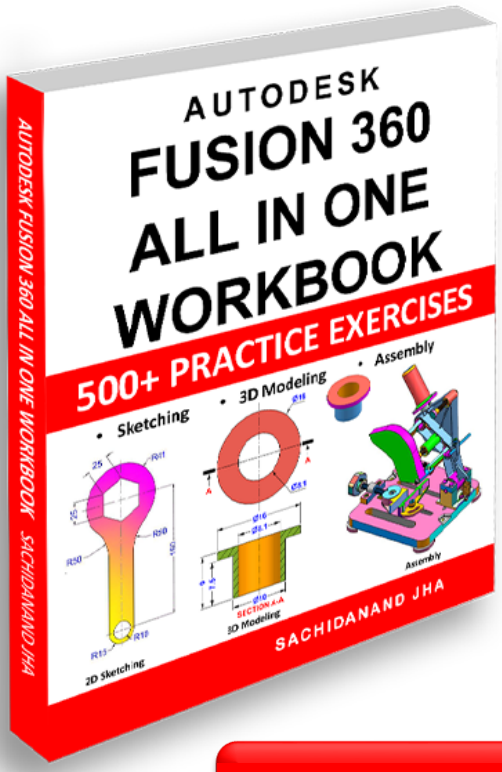

What's Included in the FUSION 360 ALL IN ONE WORKBOOK?

- ✓ 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
- ✓ Instant Download Link - Sent to Your Email After Payment
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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)