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 EDIT SHELL THICKNESS IN FUSION 360

• LEARN • • APPLY • • GROW •

Introduction

Editing shell thickness in Fusion 360 is a fundamental task for designing 3D models that meet specific strength, weight, or aesthetic requirements. Proper control over shell parameters allows for the creation of lightweight hollow objects or parts with precise wall thicknesses. Whether you're designing a case, a prototype, or a functional component, understanding how to modify shell thickness efficiently can significantly improve your workflow. In this guide, we'll explore step-by-step methods to edit shell thickness in Fusion 360, share practical tips, highlight common mistakes, and compare different approaches. This comprehensive tutorial aims to give you the confidence to manipulate shell thickness like a pro, ensuring your designs are both functional and manufacturable.

How to Edit Shell Thickness in Fusion 360

Fusion 360 offers powerful tools for creating and modifying shells. The core function involves converting solid models into hollow parts with consistent or variable wall thicknesses. Here, we'll walk through the process of editing shell thickness on existing models, covering both simple and complex cases.

1. Using the Shell Tool for Initial Creation

Before editing shell thickness, you need to understand how to apply shells initially, which sets the foundation for future modifications.

- Open your fusion model.
- Select the solid body you want to shell.
- Navigate to the **Solid** workspace if not already there.
- Click on the **Modify** dropdown menu.
- Choose **Shell**.

This tool will prompt you to specify the desired wall thickness for your hollowed-out model.

2. Setting the Original Shell Thickness

Once you've activated the Shell command:

- Click on the faces or bodies you want to shell.
- In the dialog box, enter the desired **thickness** value.
- Specify **which faces to remove**:
- All faces if you want an enclosed shell.
- Selected faces if you want partial shells or openings.
- Confirm by clicking **OK**.

This creates a uniform shell thickness across the selected faces. To modify this later, proceed to the next step.

3. Editing Shell Thickness After Creation

In Fusion 360, once a shell is created, you can adjust its thickness using different techniques depending on your modeling needs.

Method A: Direct Edit via the Timeline

- Find the **Shell** feature in the Fusion 360 timeline (bottom of the screen).
- Right-click on the **Shell** feature.
- Choose **Edit Feature**.
- In the dialog box, change the **thickness** value.
- Click **OK**.

This method updates the shell's thickness uniformly, reflecting the new value immediately.

Method B: Using the "Press Pull" Tool

- Select the hollowed-out body or the specific faces.
- Activate the **Press Pull** tool from the **Modify** menu.
- Click on the inner face(s) you wish to modify.

- Enter a new thickness value or drag to adjust dynamically.
- Confirm the changes.

Note: This method is useful for fine-tuning specific sections but may require additional cleanup.

4. Creating Variable Shell Thicknesses

For complex designs requiring different wall thicknesses in various regions:

- Use **UCS (User Coordinate System)** or **Section Analysis** to identify regions.
- Use **Split Body** to isolate specific areas.
- Apply **Shell** separately to different sections with distinct thicknesses.
- Alternatively, create **additional shells** on different faces, each with custom thickness values.

5. Practical Example: Hollowing Out a Water Bottle

Imagine you have a solid water bottle model:

- Step 1: Select the entire bottle body.
- Step 2: Use the **Shell** tool and set the initial thickness to 2 mm.
- Step 3: To make the base thinner, select the base face.
- Step 4: Use **Press Pull** to reduce thickness selectively to 1 mm.
- Step 5: Fine-tune the sidewalls to achieve a perfect balance between strength and weight.

This illustrates how to effectively modify shell thickness after initial creation for real-world applications.

Common Mistakes When Editing Shell Thickness

When working with shell modifications, certain pitfalls can hinder your progress:

- **Applying shell with zero or too low thickness**: This can produce invalid geometry or errors.
- **Not updating the timeline feature**: Failing to edit the original shell feature leaves you unable to modify the thickness later.
- **Ignoring internal geometry**: Overlooking internal features can cause issues with wall thickness or unwanted holes.
- **Using the wrong method for complex geometries**: Employing just the Shell tool without considering multiple shells or localized modifications can result in inaccuracies.

Best Practices and Pro Tips

- Always plan your shell thickness beforehand for complex parts.
- Use the **Edit Feature** option to adjust existing shells without rebuilding the model.
- For variable thicknesses, combine multiple shell features or use contouring techniques.
- When working on intricate models, create section views to visualize internal wall thickness.
- Regularly save incremental versions of your file before making major adjustments.

Comparing Different Approaches to Shell Thickness Editing

Method	Pros	Cons	Best Use Case

Editing the Timeline Shell Feature	Simple, quick for uniform changes	Cannot create variable thickness	Simple models with uniform shell
Press Pull on Inner Faces	Fine control, localized adjustments	Can be time-consuming for complex parts	Fine-tuning specific areas
Multiple Shells	Precise control over different regions	More complex setup	Parts requiring variable wall thicknesses

Conclusion

Mastering how to edit shell thickness in Fusion 360 empowers you to create optimized, realistic, and functional models. Whether you're applying a simple uniform shell or designing complex parts with variable thicknesses, understanding these methods allows you to adapt quickly to design challenges. Always plan your shell features carefully, use feature editing for flexibility, and employ best practices to avoid common mistakes. With these skills, you'll enhance your design efficiency and produce high-quality, manufacturable parts.

FAQ

1. How can I change the wall thickness of an existing shell in Fusion 360?

Ans : You can right-click the original Shell feature in the timeline and select *Edit Feature* to modify the wall thickness.

2. Is it possible to create shells with different thicknesses in the same component?

Ans : Yes, by applying multiple shell features to different regions or faces with distinct thickness settings.

3. Can I modify shell thickness after exporting the model?

Ans : No, shell thickness adjustments should be made within Fusion 360 before exporting; post-export modifications are limited.

4. How do I create a shell with variable thickness in Fusion 360?

Ans : Use multiple shell features for different regions or utilize the Press Pull tool on specific faces to fine-tune thicknesses.

5. What are common issues when editing shell thickness?

Ans : Common issues include invalid geometry with very low thicknesses, forgetting to update the timeline feature, and internal geometry conflicts.

6. Is there a way to visualize wall thickness in Fusion 360?

Ans : Yes, use section analysis and visualize internal regions to assess wall thickness.

7. What is the best approach for designing hollow objects with precise shell thickness?

Ans : Start with the Shell tool for uniform thickness, then use the Edit Feature or Press Pull tools for localized adjustments to refine the design.

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.

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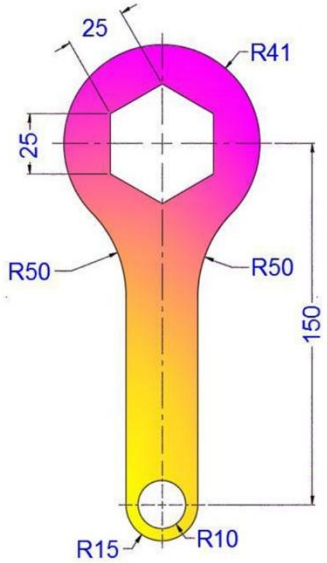
- Sketching
- 3D Modeling
- Assembly



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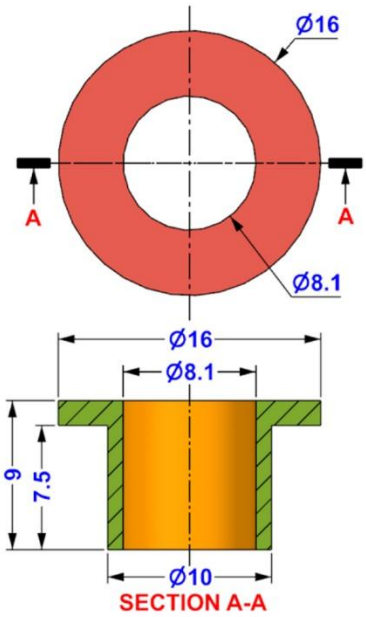
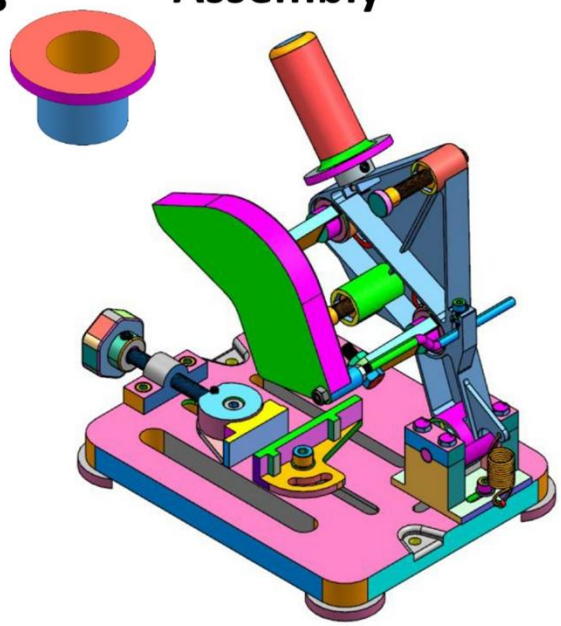


3D Modeling



Assembly

SACHIDANAND JHA

AUTODESK

FUSION 360 ALL IN ONE WORKBOOK

500+ PRACTICE EXERCISES

2D Sketching • 3D Modeling • Assembly Drawings

SACHIDANAND JHA

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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.

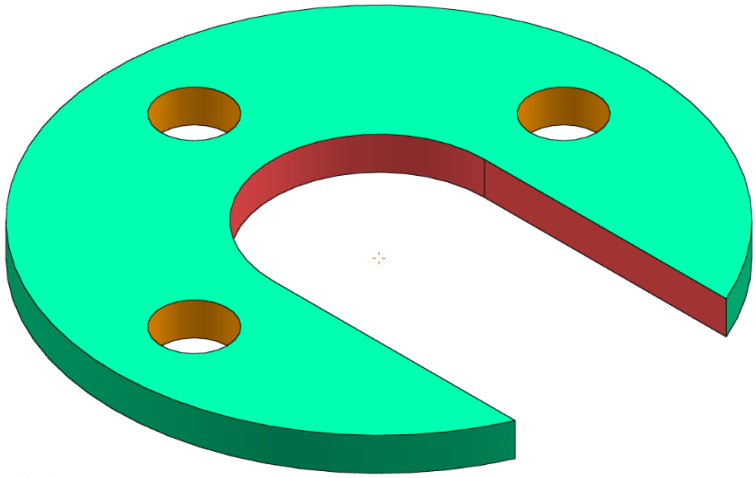
Tips for Best Use:

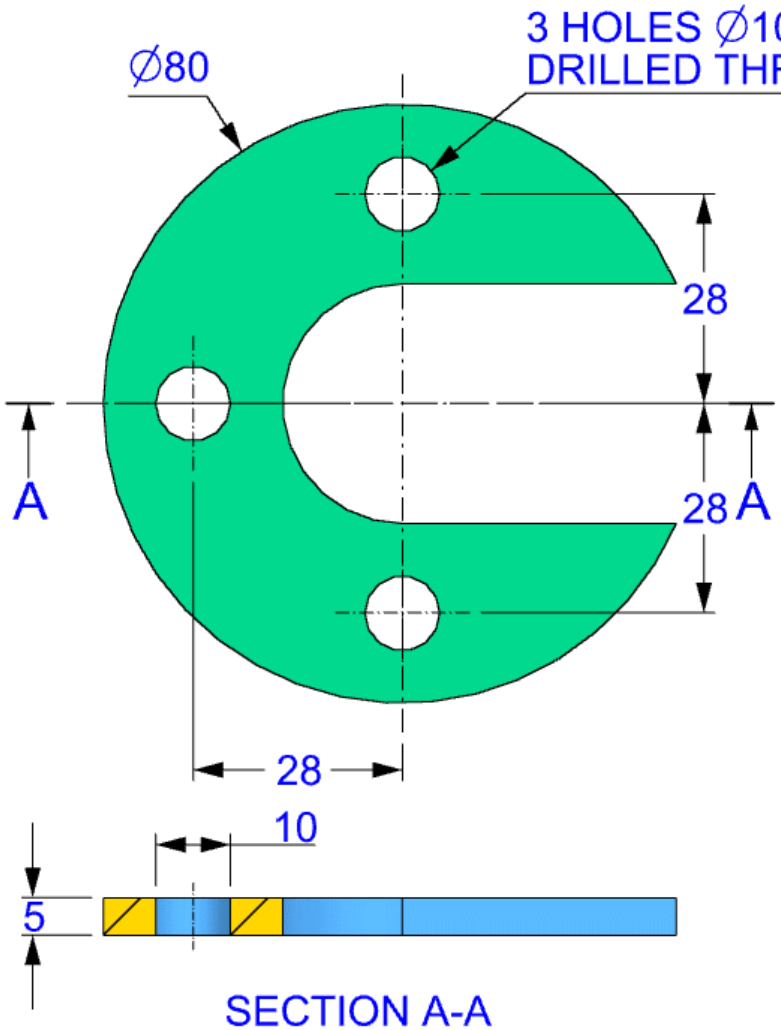
- Complete the exercises in order, or jump to any skill level you prefer.
- All dimensions are in millimeters.
- Where dimensions are missing, apply logic or practice estimation.
- This book is ideal for both students and professionals preparing for industry design work.

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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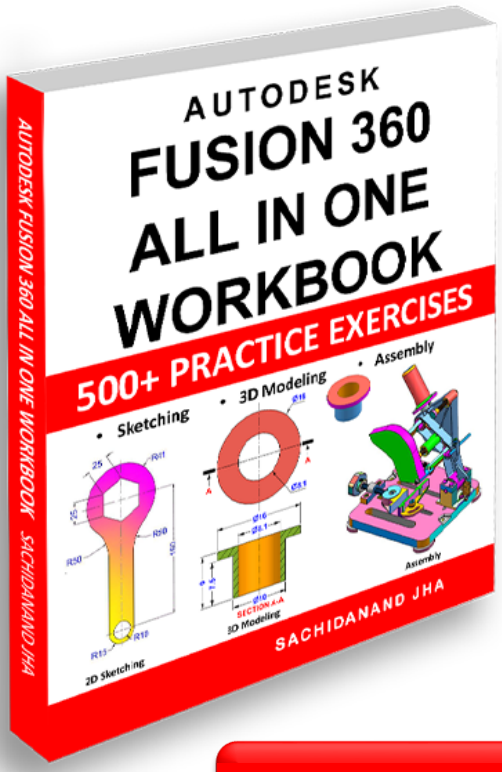

What's Included in the FUSION 360 ALL IN ONE WORKBOOK?

- ✓ 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
- ✓ Get All Assembly Exercises in .STP file
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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)