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 CHANGE THICKNESS OF SOLID IN FUSION 360

• LEARN • • APPLY • • GROW •

Introduction

Changing the thickness of a solid body in Fusion 360 is a fundamental task that allows designers to customize and refine their models. Whether you're creating prototypes, manufacturing parts, or adjusting a design for specific strength requirements, mastering how to modify thickness efficiently can save time and improve accuracy. This guide provides step-by-step instructions, best practices, and tips to help you easily alter the thickness of solid models in Fusion 360. If you want to streamline your workflow and achieve precise results, understanding how to change the thickness of a solid in Fusion 360 is essential.

How to Change the Thickness of a Solid in Fusion 360

Adjusting the thickness of a solid can be approached in several ways, depending on whether you want to uniformly change its entire thickness or modify specific parts. Below, we discuss the most effective methods to do this in Fusion 360.

1. Using the Scale Tool for Uniform Thickness Adjustment

The Scale tool is a quick way to uniformly resize your solid, including its thickness.

- Open your Fusion 360 model
- Select the solid body you wish to resize
- Go to the “Modify” menu
- Click on “Scale”
- In the dialog box:
- Choose the "Solid" option
- Select the object in the canvas if not already selected
- Specify the scale factor
- For changing thickness, use a uniform scale (e.g., 1.2 to increase by 20%)
- Click "OK" to apply the change

Practical Tip: For precise control over thickness, use the scale factor based on the ratio of the desired thickness to the original.

2. Using the Press Pull Tool to Manually Adjust Thickness

The Press Pull tool allows you to increase or decrease the thickness by dragging existing faces.

- Select the face(s) representing the thickness you want to modify
- Activate the “Press Pull” feature from the “Modify” menu
- Click on the face you want to change
- Drag inward or outward to adjust the thickness manually
- Input the exact distance if precise measurement is necessary
- Confirm the operation

Pro Tip: Use this method when you need to fine-tune specific areas of your solid, such as increasing wall thickness or creating offsets.

3. Using the Extrude Tool for Precise Thickness Adjustment

Extrude is ideal when creating new features or modifying existing bodies based on sketches or profiles.

- Select the profile or face of the solid
- From the “Create” menu, choose “Extrude”
- Set the direction of extrusion (typically perpendicular)
- Input the new desired thickness value
- Extend or cut the material based on your design needs
- Click “OK” to finalize

Real-World Example: Modifying the thickness of a plate or chassis to meet structural specifications.

4. Editing the Body in the Solid Modeling Environment

Fusion 360 allows you to directly modify the solid body through editing features.

- Right-click on the body in the Browser panel
- Select “Edit Form” or “Edit Feature” (depending on the approach)
- Use push/pull, scale, or move tools to modify existing geometry
- Confirm changes and ensure the updated thickness is accurate

5. Using the Offset Face Tool for Uniform Thickness Changes

The Offset Face tool creates an offset on selected faces, useful for uniform thickness adjustments across entire surfaces.

- Select the face(s) to adjust
- Go to the “Modify” menu
- Choose “Offset Face”
- Set the offset distance (positive for outward, negative for inward)
- Preview the change to ensure correct thickness adjustment
- Click “OK” to apply

Best Practice: Use this method when you need to uniformly reduce or increase the thickness of a shell or face.

Common Mistakes and How to Avoid Them

- **Not selecting the correct faces or bodies:** Always double-check your selections before applying modifications to avoid unintended changes.
- **Ignoring units and measurements:** Use precise input or refer to your model’s dimensions for accuracy.

- **Overlooking the impact of thickness change on other features:** When increasing thickness, ensure that the change doesn’t interfere with other parts or assemblies.
- **Using non-uniform scaling where uniform thickness is needed:** For consistent thickness, prefer tools like Offset Face or specific extrusion rather than free-form scaling.

Practical Examples of Changing Thickness

- Adjusting the wall thickness of a 3D-printed enclosure for durability.
- Increasing the thickness of a base plate to improve load-bearing capacity.
- Reducing the thickness of a prototype shell to save material costs.
- Customizing key features like ribs or reinforcement webs for strength.

Pro Tips and Best Practices

- Always duplicate your original model before making major adjustments; this preserves an unaltered version.
- Use precise measurements and dimension constraints when modifying thickness to ensure consistency.
- Combine multiple methods—like Offset Face for general adjustments and Press Pull for detail work—to get better control.
- Regularly analyze your model with tools like Section Analysis to verify the changes visually.
- Remember to keep your Fusion 360 software updated for access to the latest features and improvements.

Comparing Methods for Changing Thickness

Method	Pros	Cons	Best Use Case
Scale Tool	Fast, uniform resizing	Less control over specific faces	Overall proportional size changes
Press Pull	Precise, manual adjustments	Time-consuming for complex models	Fine-tuning individual areas
Extrude	Accurate, based on profiles	Requires sketches or profiles	Creating or modifying features

Offset Face	Uniform face adjustments	Limited to planar faces	Shelling or reducing wall thickness
Direct Editing	Flexible, direct manipulation	Might distort geometry if not careful	Quick edits on complex bodies

Conclusion

Changing the thickness of a solid in Fusion 360 is a fundamental skill that enhances your ability to customize parts accurately. Whether you're adjusting entire bodies with scaling or fine-tuning specific faces with offset or press pull, the right method depends on your project requirements. Practice and familiarity with these tools will streamline your workflow, improve your precision, and ultimately lead to better-designed models. Remember to always double-check your modifications and use the appropriate method for your specific task.

FAQ

1. How can I uniformly change the thickness of a solid in Fusion 360?

Ans : Use the Offset Face tool or the Scale feature for uniform adjustments across the whole solid.

2. What is the best way to modify thickness in complex assemblies?

Ans : Use the Press Pull tool for specific faces or features, and combine it with the Offset Face tool for overall consistency.

3. Can I change the thickness of a shell in Fusion 360?

Ans : Yes, using the Offset Face feature is ideal for shell bodies, allowing you to increase or decrease wall thickness.

4. How do I ensure my thickness modifications don't interfere with other components?

Ans : Use the Section Analysis tool to visualize internal features and verify clearances after modifying thickness.

5. Is there a way to change thickness nondestructively?

Ans : Using parametric features like Offset Face or Emboss can allow for adjustments without permanently altering original geometry.

6. What precautions should I take before changing thickness in Fusion 360?

Ans : Always save a copy of your original model, and ensure you have accurate measurements to avoid errors.

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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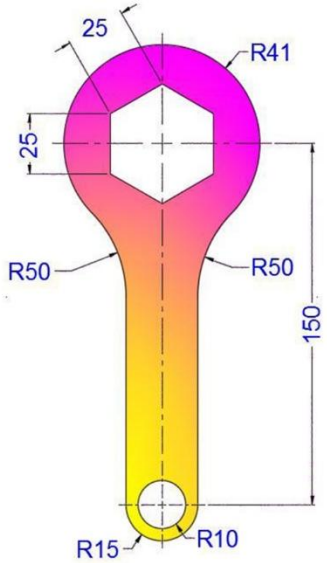
- Sketching
- 3D Modeling
- Assembly



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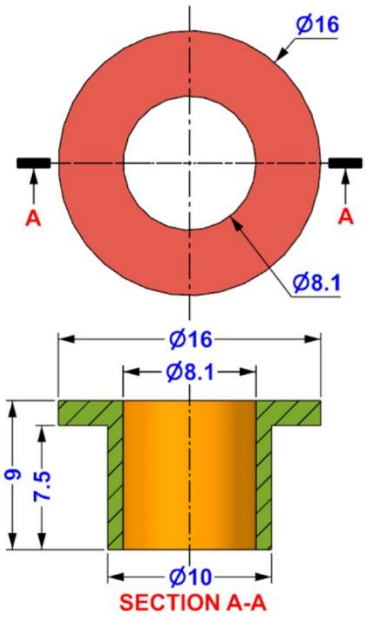
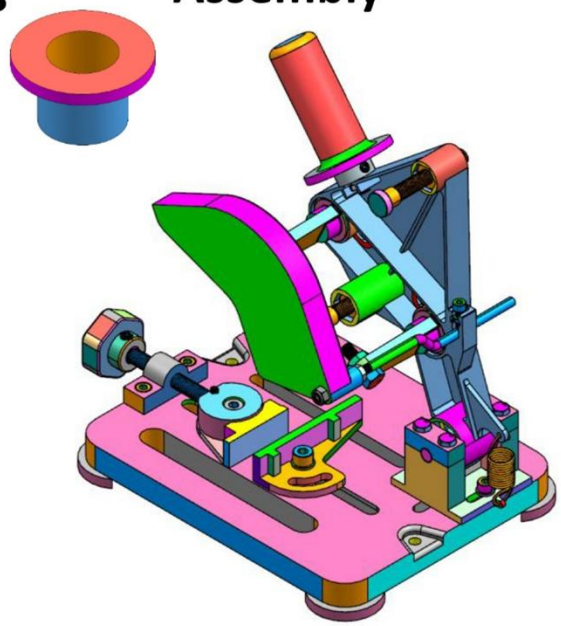


3D Modeling



Assembly

SACHIDANAND JHA

AUTODESK

FUSION 360 ALL IN ONE WORKBOOK

500+ PRACTICE EXERCISES

2D Sketching • 3D Modeling • Assembly Drawings

SACHIDANAND JHA

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.

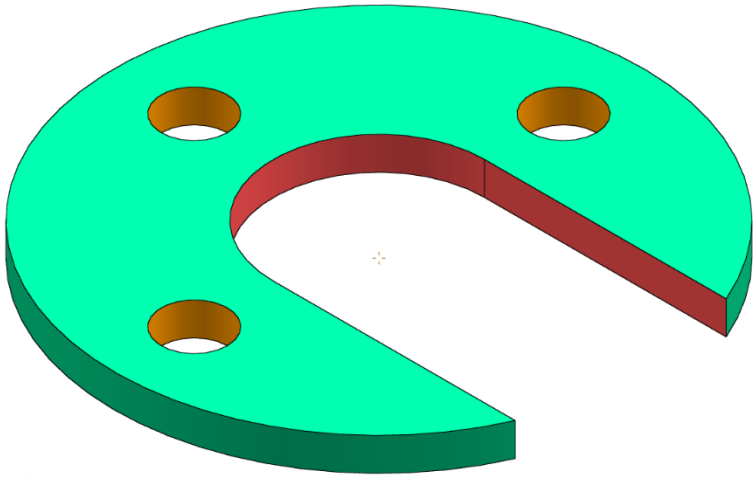
Tips for Best Use:

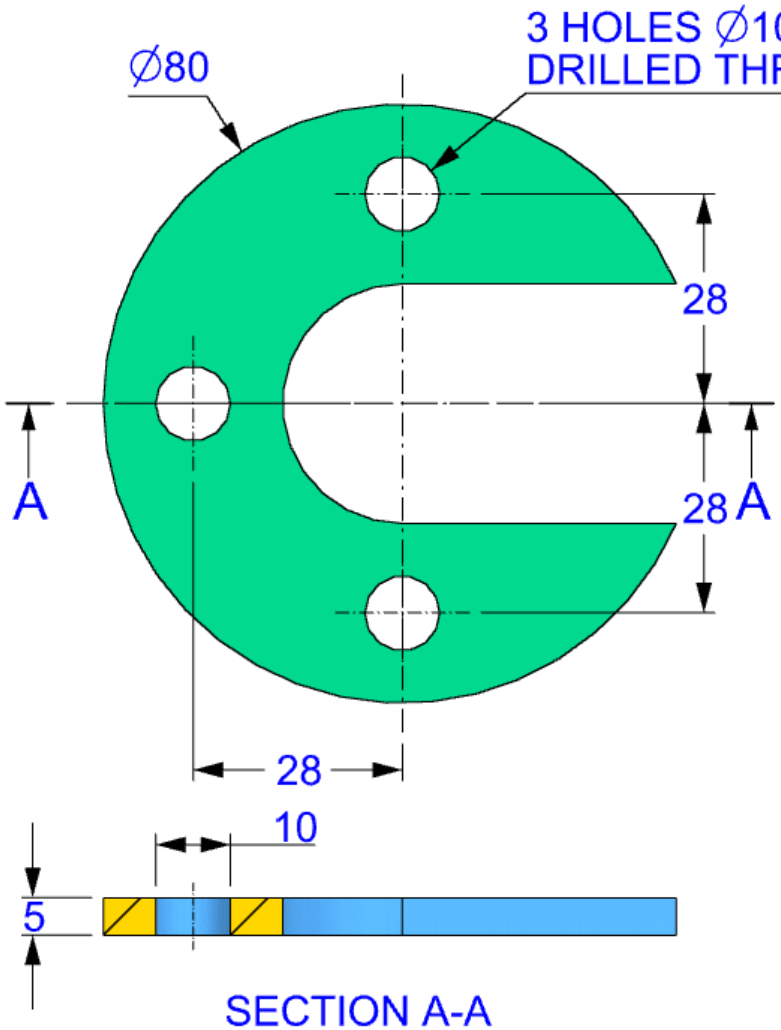
- Complete the exercises in order, or jump to any skill level you prefer.
- All dimensions are in millimeters.
- Where dimensions are missing, apply logic or practice estimation.
- This book is ideal for both students and professionals preparing for industry design work.

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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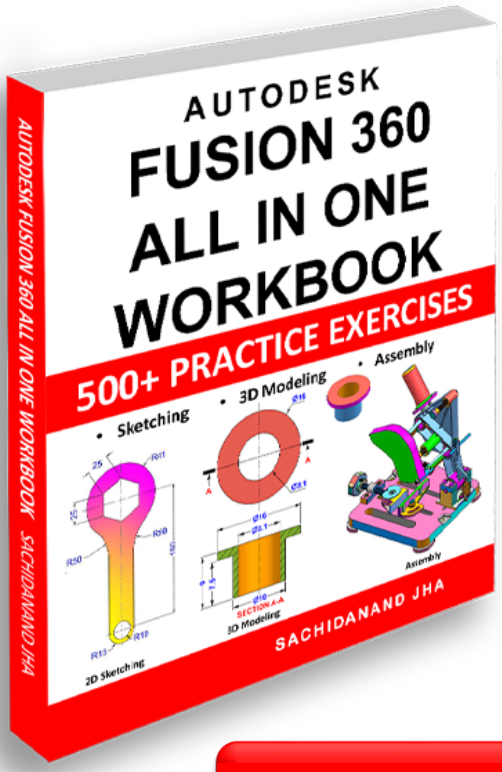

What's Included in the FUSION 360 ALL IN ONE WORKBOOK?

- ✓ 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)