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.

Disclaimer:

— CADIN360 Team

HOW TO EDIT SOLID WITHOUT EDITING SKETCH IN FUSION 360

• LEARN • • APPLY • • GROW •

Introduction

Editing a solid in Fusion 360 without modifying its original sketch is a common requirement for many designers and engineers. Whether you're refining a complex shape or making minor adjustments, knowing how to efficiently edit solids without altering sketches can save time and preserve design intent. This guide provides a clear, step-by-step process for editing solids directly, along with tips, best practices, and troubleshooting advice to enhance your CAD workflow.

How to Edit Solid Without Editing Sketch in Fusion 360

Fusion 360 offers numerous powerful tools that allow you to refine and modify your 3D models directly, bypassing the need to edit sketches. This flexibility is especially useful when you want to maintain a clean sketch environment or when sketches are fully constrained and finalized.

1. Understanding When to Edit the Solid Directly

Before jumping into editing, it's important to recognize scenarios where editing the solid directly is advantageous:

- Fine-tuning a part's shape after sketch-based features are complete.
- Making adjustments to complex bodies without returning to sketches.
- Correcting geometric inaccuracies that are easier to modify at the solid level.
- Applying modifications to imported geometry where sketches may not exist.

2. Accessing the Edit Tools in Fusion 360

Fusion 360 provides a suite of tools explicitly designed for solid editing:

- **Move/Copy**: To shift, rotate, or duplicate parts.
- **Press Pull**: To extend or cut into existing faces.

- **Scale**: To uniformly or non-uniformly resize parts.
- **Combine**: To join, cut, or intersect bodies.
- **Fillet and Chamfer**: To modify edges for smoothness or angles.
- **Shell**: To hollow out parts.
- **Split Body**: To divide a solid into multiple parts.

3. Step-by-Step: Editing a Solid Without Sketch Modification

Here's a practical guide to editing a solid directly:

Step 1: Selecting the Body

- Open your Fusion 360 project.
- In the Browser panel, locate the body you want to edit.
- Click on the body to select it.

Step 2: Using the Move Tool

- Go to the **Solid** tab on the toolbar.
- Select **Modify** > **Move / Copy** .
- In the dialog box, choose the type of move:
- **Free Move** for arbitrary repositioning.
- **Translate** for directional shifts.
- **Rotate** to spin the body around an axis.
- Use the manipulator arrows, planes, or free move sliders to adjust positioning.
- Confirm the move with **OK**.

Step 3: Editing Faces with Press Pull

- Select **Modify** > **Press Pull** .

- Click on a face you wish to modify.
- Drag the face to new position or enter precise values.
- Use the dialog box options to control the amount of extrusion or cut.
- Confirm with **OK**.

Step 4: Reshaping with Scale

- Choose **Modify** > **Scale** .
- Select the body or specific faces.
- Use the scale manipulator or input exact ratios to resize.
- Apply the changes directly without affecting sketches.

Step 5: Cutting or Combining Bodies

- To cut or join bodies:
- Use **Combine** under **Modify** .
- Select the target body and the tool body.
- Choose the operation: **Join**, **Cut**, or **Intersect**.
- Adjust position if necessary before confirming.

4. Real-World Example: Reshaping a Mechanical Part

Suppose you have a solid block with holes, and you want to adjust the outer dimensions without changing the holes' placement:

- Use **Move / Copy** to reposition the entire block.
- Select **Press Pull** on the outer faces to resize the block.
- If needed, use **Fillet** for rounded edges.

- Use **Split Body** to add separate sections without affecting features created through sketches.

5. Common Mistakes to Avoid

- **Modifying sketches inadvertently**: Ensure you are working on bodies and not sketches.
- **Not selecting the correct body**: Always verify which body is active before editing.
- **Applying destructive modifications without backups**: Save versions or duplicates before large edits.
- **Using inappropriate tools for complex changes**: For intricate redesigns, consider editing sketches or parametric features.

6. Best Practices and Pro Tips

- **Use the timeline**: Keep track of modifications to revert if needed.
- **Parametrize your design**: Even when editing solids directly, use parameters for features to maintain control.
- **Combine editing methods**: Use both solid editing and sketch adjustments for optimal results.
- **Leverage shortcuts**: Familiarize yourself with hotkeys for faster access.

Comparing Solid Editing and Sketch Editing in Fusion 360

Aspect	Solid Editing	Sketch Editing

Flexibility	Directly modify bodies, faces, and features	Change design intent and geometry at the sketch level
Use Cases	Minor tweaks, complex geometrical adjustments	Creating or redesigning features from scratch
Risks	Can be destructive if not careful	Maintains parametric control but may require retracing steps
Best Practice	Use for post-creation adjustments	Use during initial design phase or major redesigns

Conclusion

Editing a solid without editing the sketch in Fusion 360 is a fundamental skill that enhances your efficiency and flexibility in CAD design. Whether you're fine-tuning parts, correcting imported geometry, or reshaping components, knowing how to directly manipulate your models allows for faster iteration and more precise adjustments. Remember to use the appropriate tools, follow best practices, and always save backup versions to keep your workflow smooth and error-free.

FAQ

1. How can I edit a solid in Fusion 360 without affecting the original sketch?

Ans : Use direct edit tools like Move/Copy, Press Pull, Scale, and Combine to modify the solid independently from the sketch.

2. Can I change the thickness of a part without editing the sketch?

Ans : Yes, using the Press Pull tool on faces allows you to modify thickness directly without modifying the sketch.

3. Is it possible to resize a body proportionally in Fusion 360?

Ans : Yes, the Scale tool enables proportional rescaling of bodies or specific features.

4. What's the best way to make precise adjustments to a solid object?

Ans : Use numeric input within tools like Move, Scale, or Press Pull to make exact, measurable modifications.

5. How do I ensure I don't ruin my original design while editing bodies directly?

Ans : Save a duplicate or version of your design before making significant modifications to safeguard your original model.

6. Can I edit imported geometry directly without creating sketches?

Ans : Yes, imported geometry can be directly modified using solid editing tools without needing to recreate sketches.

7. Are there limitations to editing solids without sketches?

Ans : Complex parametric adjustments or feature-specific modifications may still require updates in the sketches or features for better control.

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

FUSION 360 ALL IN ONE WORKBOOK

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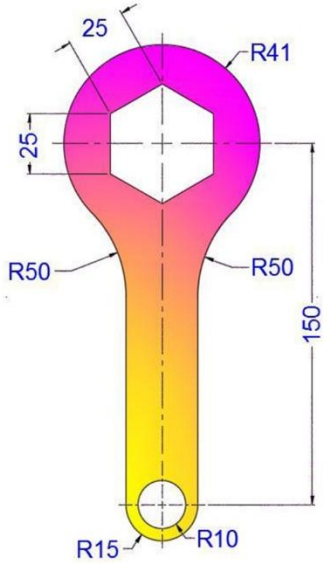
- Sketching
- 3D Modeling
- Assembly



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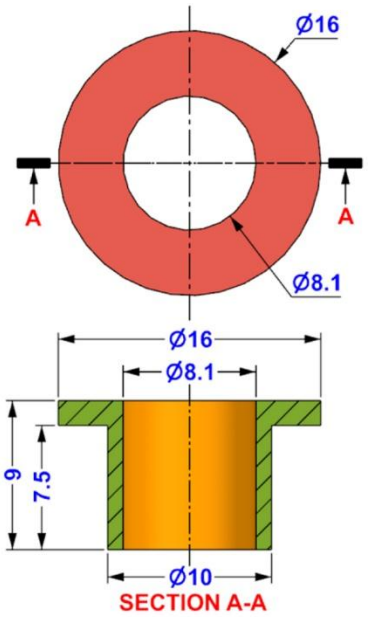
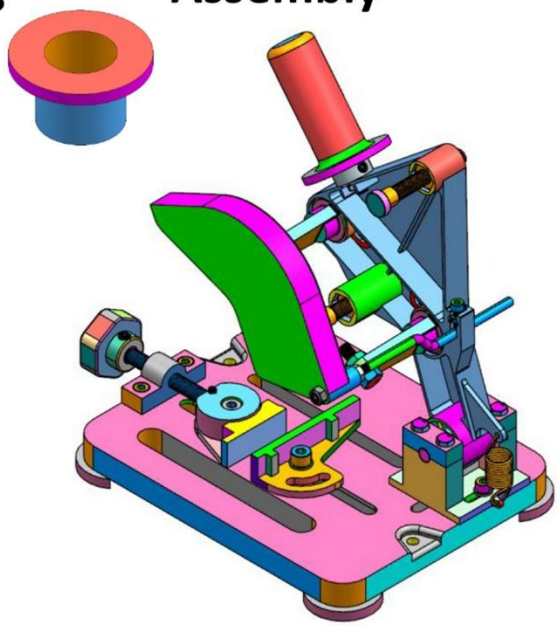


3D Modeling



Assembly

SACHIDANAND JHA

AUTODESK

FUSION 360 ALL IN ONE WORKBOOK

500+ PRACTICE EXERCISES

2D Sketching • 3D Modeling • Assembly Drawings

SACHIDANAND JHA

Dear Reader,

Thank you for choosing the AUTODESK FUSION 360 ALL IN ONE WORKBOOK. This book is part of the CADIN360° learning series, created to help engineers, students, and professionals master Fusion 360 through structured and practical exercises.

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:

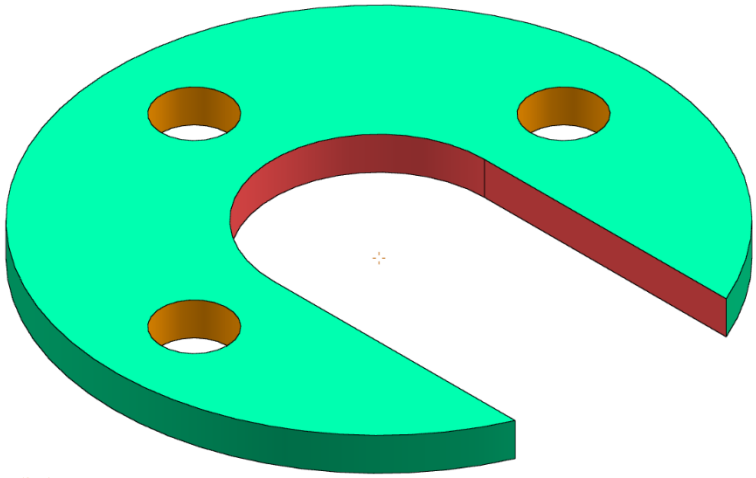
- Complete the exercises in order, or jump to any skill level you prefer.
- All dimensions are in millimeters.
- Where dimensions are missing, apply logic or practice estimation.
- This book is ideal for both students and professionals preparing for industry design work.

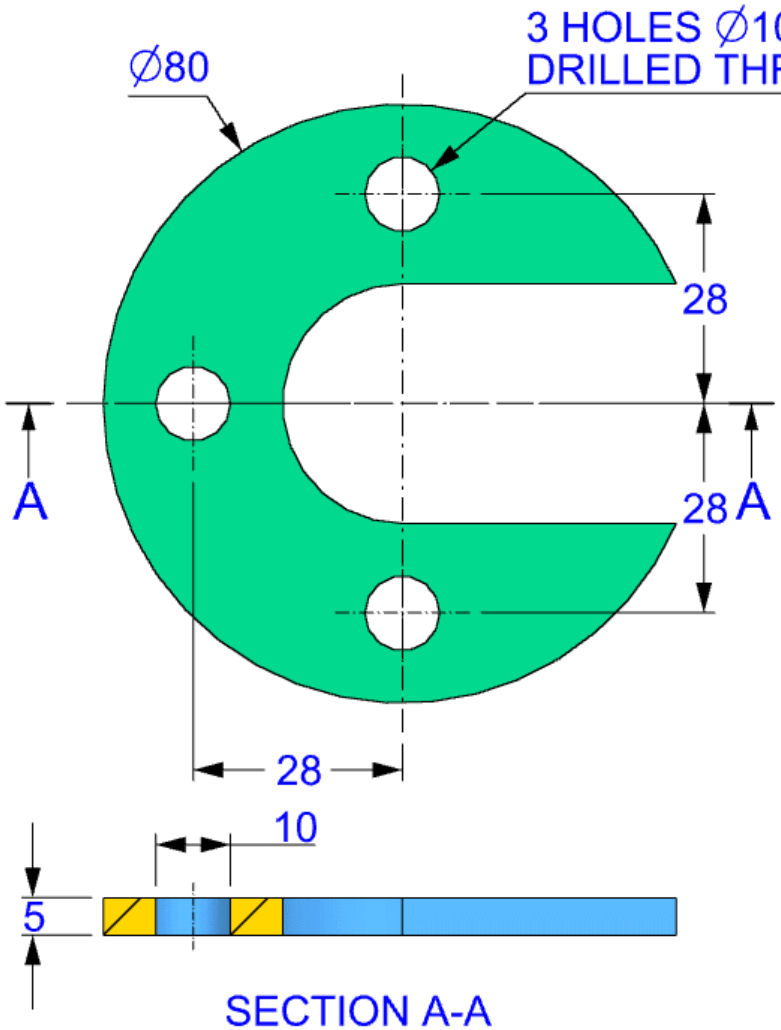
Note:

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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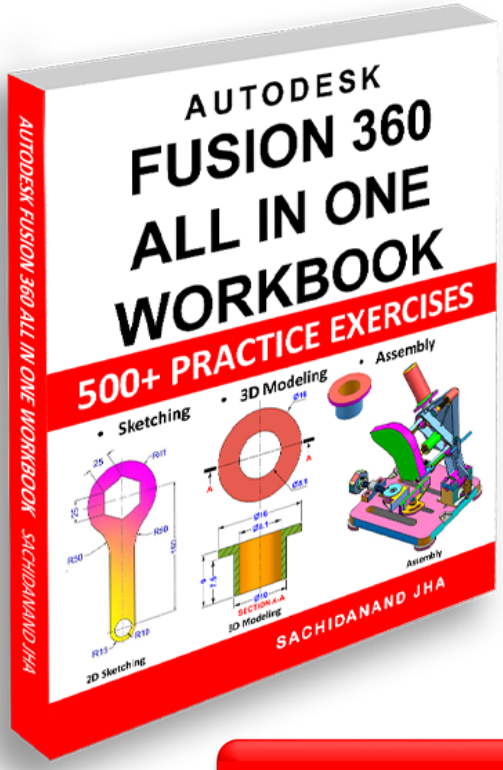

What's Included in the FUSION 360 ALL IN ONE WORKBOOK?

- ✓ 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
- ✓ 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)