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 SWITCH FROM SKETCH TO SOLID WORKSPACE IN FUSION 360

• LEARN • • APPLY • • GROW •

Introduction

Switching from Sketch workspace to Solid workspace in Fusion 360 is a common task for designers and engineers aiming to transition from 2D sketching to solid modeling. While Sketch workspace is ideal for creating 2D profiles, Solid workspace is essential for building three-dimensional models with precise features. Understanding how to seamlessly make this transition enhances your workflow, boosts productivity, and opens up new design possibilities. In this guide, you'll learn step-by-step how to switch from Sketch to Solid workspace in Fusion 360, along with practical tips, common mistakes to avoid, and real-world examples to optimize your modeling process.

Understanding the Difference Between Sketch and Solid Workspaces

Before diving into the transition process, it's vital to understand the distinction between Sketch and Solid workspaces:

- **Sketch Workspace:** Focused on creating 2D profiles and sketches using lines, arcs, circles, and other geometry. These sketches serve as the foundation for making 3D models.
- **Solid Workspace:** Used for generating 3D models by extruding, revolving, or combining sketches into three-dimensional shapes.

A clear understanding of these workspaces helps in planning your design process and ensures smooth workflows.

How to Switch from Sketch to Solid Workspace in Fusion 360

Switching from Sketch to Solid workspace involves creating 3D features based on your sketches. Follow these precise steps to make this transition efficiently:

1. Finish or Exit Sketch Mode

- Complete your sketch if you are still in the sketch environment.
- Click on the **Finish Sketch** button in the Toolbar, typically located at the top. This action exits you from Sketch workspace, making it ready for 3D modeling.

2. Select Your Sketch Profile for 3D Operations

- Click on the sketch geometry in the canvas or select the profile in the Browser panel.
- Ensure the profile is closed and clean for predictable 3D operations.

3. Switch to the Solid Workspace

- At the top of Fusion 360, click on the **Design** dropdown menu.
- Select **Solid** from the workspace options if needed. Usually, Fusion 360 maintains the active workspace, but if you need to switch between environments, this step is crucial.
- Alternatively, the toolbar automatically switches to Solid tools once you start creating features, so explicit switching is often unnecessary.

4. Create 3D features from your Sketch

- Use tools like **Extrude**, **Revolve**, **Sweep**, or **Loft** to convert your 2D sketch profiles into 3D models:
- For example, select your closed profile.
- Click on the **Create** menu.
- Choose **Extrude**.
- Define height and direction parameters.
- This process effectively “transfers” your sketch into a solid.

5. Refine Your Solid Model

- Use other tools such as fillets, chamfers, shells, and patterns to refine your solid model.

- Continue transitioning from sketch-based profiles to complete 3D parts using solid modeling techniques.

Practical Example: Creating a Solid Block from a Sketch

Imagine you've drafted a simple rectangle sketch for a cube base:

1. **Finish your rectangle sketch.**
2. **Select the entire profile.**
3. **Click Create > Extrude.**
4. **Type in the depth for your solid, e.g., 50 mm.**
5. **Confirm the extrusion, and you'll have a solid block directly from your initial sketch.**

This straightforward example demonstrates how easy it is to progress from sketch to solid.

Common Mistakes When Switching from Sketch to Solid Workspace

To ensure a smooth workflow, be aware of typical errors:

- **Leaving open sketches:** Ensure profiles are closed; open profiles won't extrude properly.
- **Incorrect selection:** Select only the closed profile; selecting unnecessary geometry can cause errors.
- **Not completing the sketch:** Don't attempt to extrude or create features before finishing the sketch.
- **Confusing the active workspace:** Make sure you are in the correct workspace; although Fusion 360 maintains context, switching views may be necessary in complex models.

Best Practices and Tips for Seamless Transition

- **Organize your sketches:** Use the Browser to rename and organize sketches for easier identification.
- **Sketch in the correct plane:** Draw your sketches on the appropriate XY, YZ, or ZX plane based on your design.
- **Use construction planes:** When designing complex parts, create multiple planes to draw sketches for different sections.
- **Validate profiles:** Use the **Inspect > Find Intersections** tool to check for gaps or open profiles.
- **Leverage parametric inputs:** When extruding or revolving, input exact dimensions to create precise solids.

Comparing Sketch and Solid Workspaces: A Quick Summary

Feature	Sketch Workspace	Solid Workspace
Purpose	To create 2D profiles	To develop 3D models based on profiles

Primary tools	Line, circle, arc, rectangle, spline, etc.	Extrude, Revolve, Sweep, Loft
Output	Closed or open profiles	3D solid bodies
Workflow focus	Design 2D geometry	Transform 2D profiles into 3D shapes
Transition process	Finish sketch, then extrude/revolve into solid	Use 3D features to refine model

Understanding this distinction helps in planning your modeling process effectively.

Pro Tips for Advanced Users

- Use **Pattern** and **Mirror** tools in the Solid workspace to copy features efficiently.
- Convert multiple sketches into complex assemblies using joints and constraints.
- Use **Parametric Equations** for precise control over dimensions during sketching.
- Save component states often to revert changes if necessary.

Conclusion

Transitioning from Sketch to Solid workspace in Fusion 360 is a fundamental skill that enhances your ability to create complex 3D designs efficiently. By following these step-by-step instructions—finishing your sketch, selecting profiles, and applying extrusion or other 3D features—you can seamlessly convert 2D sketches into detailed 3D models. Remember to organize your sketches, verify closed profiles, and leverage best practices to avoid common pitfalls. Mastering this workflow empowers you to design smarter, faster, and with greater precision, unlocking the full potential of Fusion 360's powerful modeling capabilities.

FAQ

1. How do I convert a sketch into a solid in Fusion 360?

Ans: Finish the sketch, select the profile, then use the **Extrude** tool or other 3D features to convert it into a solid body.

2. Can I edit my sketch after creating a solid?

Ans: Yes, you can double-click the sketch in the Browser to edit it, and then update your features accordingly.

3. What's the best way to create a complex 3D shape from a sketch?

Ans: Use a combination of **Extrude**, **Revolve**, **Sweep**, and **Loft** features in the Solid workspace based on your sketches.

4. How do I fix open or non-closed profiles that won't extrude?

Ans: Use the **Inspect > Find Intersections** tool to identify gaps and close open profiles by editing your sketch.

5. Do I need to switch workspaces manually when modeling in Fusion 360?

Ans: Not necessarily; Fusion 360 maintains context, but switching to the correct workspace or toolset ensures proper modeling workflows.

6. How can I ensure my sketch is suitable for 3D operations?

Ans: Make sure your sketch profiles are fully closed, clean, and precisely dimensioned before attempting to extrude or revolve.

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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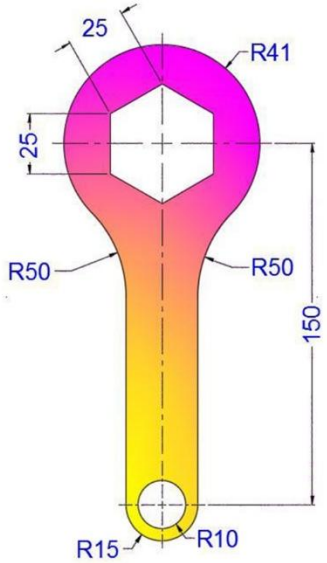
- Assembly



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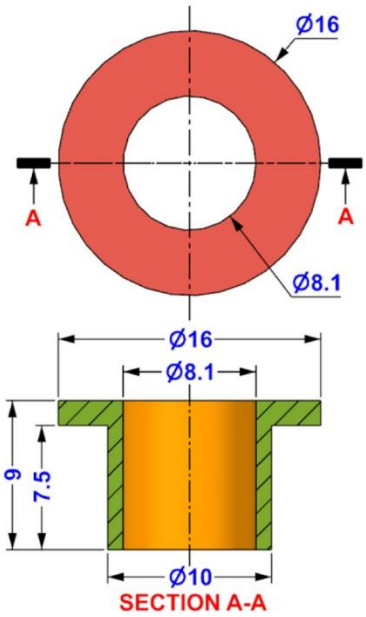
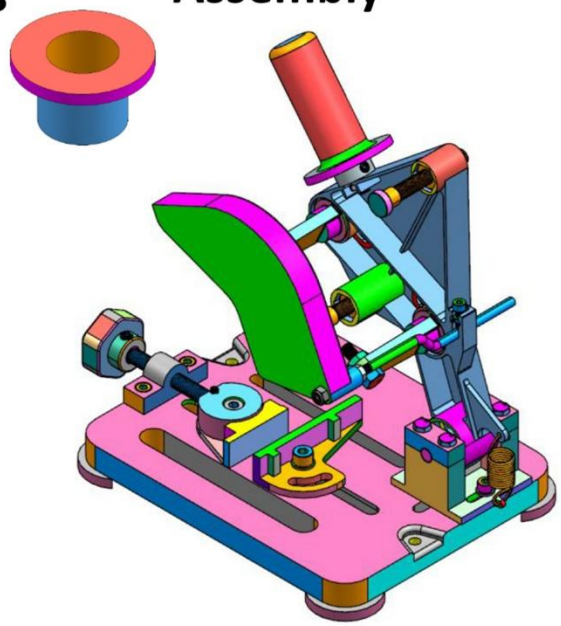


3D Modeling



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

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.

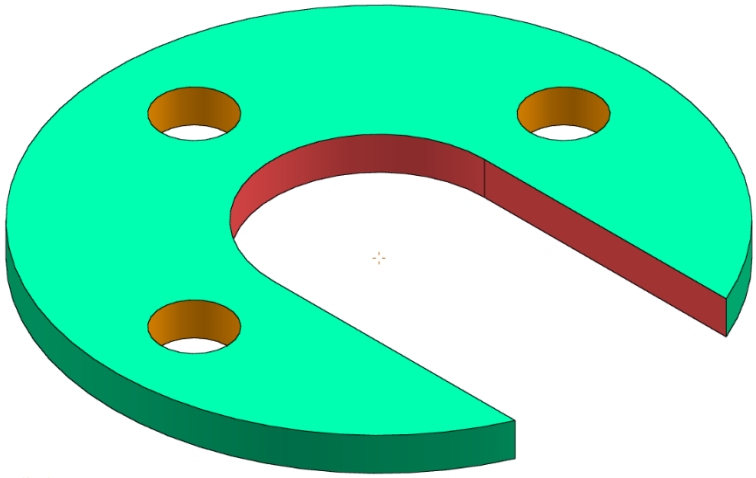
Tips for Best Use:

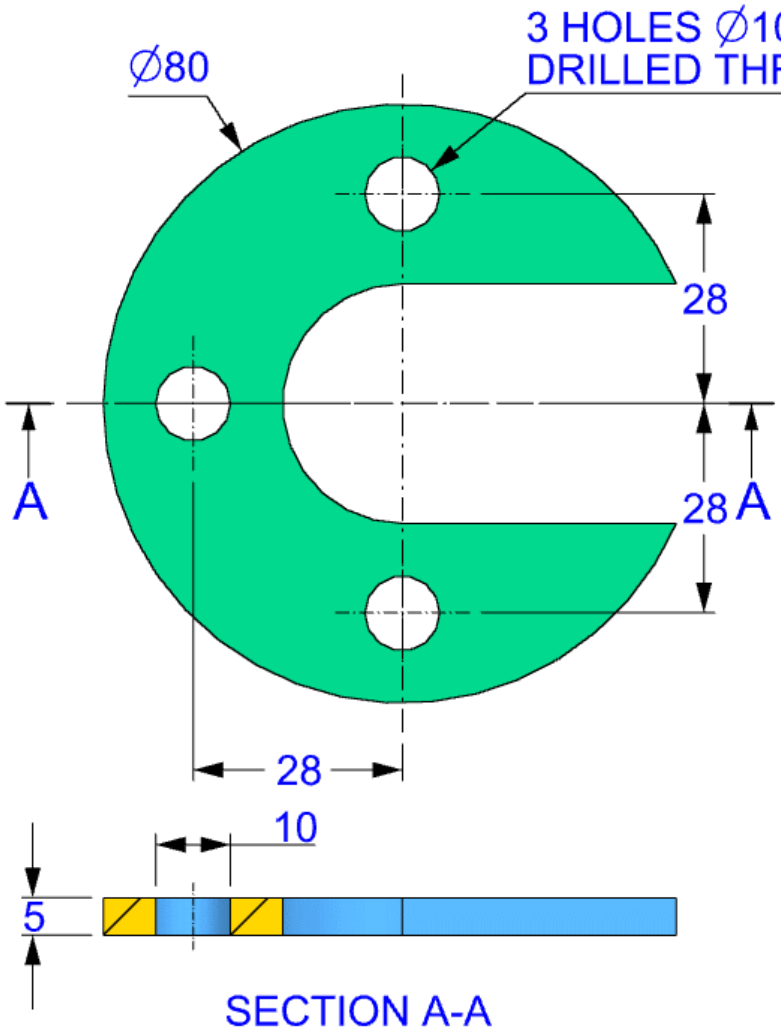
- Complete the exercises in order, or jump to any skill level you prefer.
- All dimensions are in millimeters.
- Where dimensions are missing, apply logic or practice estimation.
- This book is ideal for both students and professionals preparing for industry design work.

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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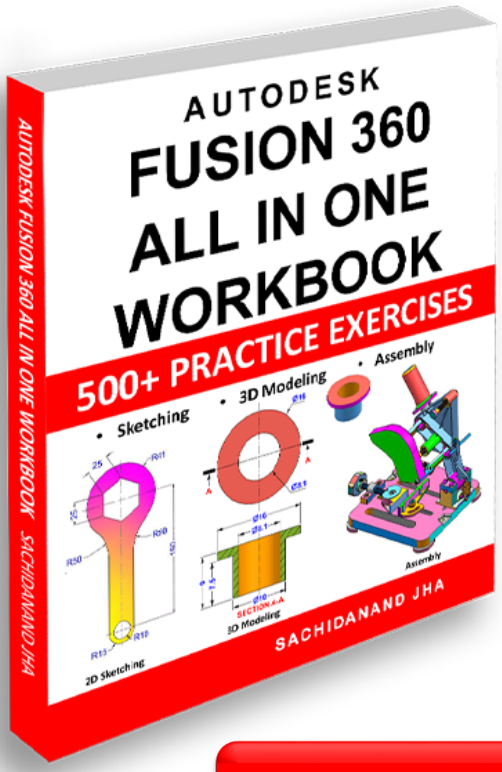

What's Included in the FUSION 360 ALL IN ONE WORKBOOK?

- ✓ 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
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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)