

AUTODESK FUSION 360

2026

BLOG

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Learning Tutorials

A Note to Our Readers

2026

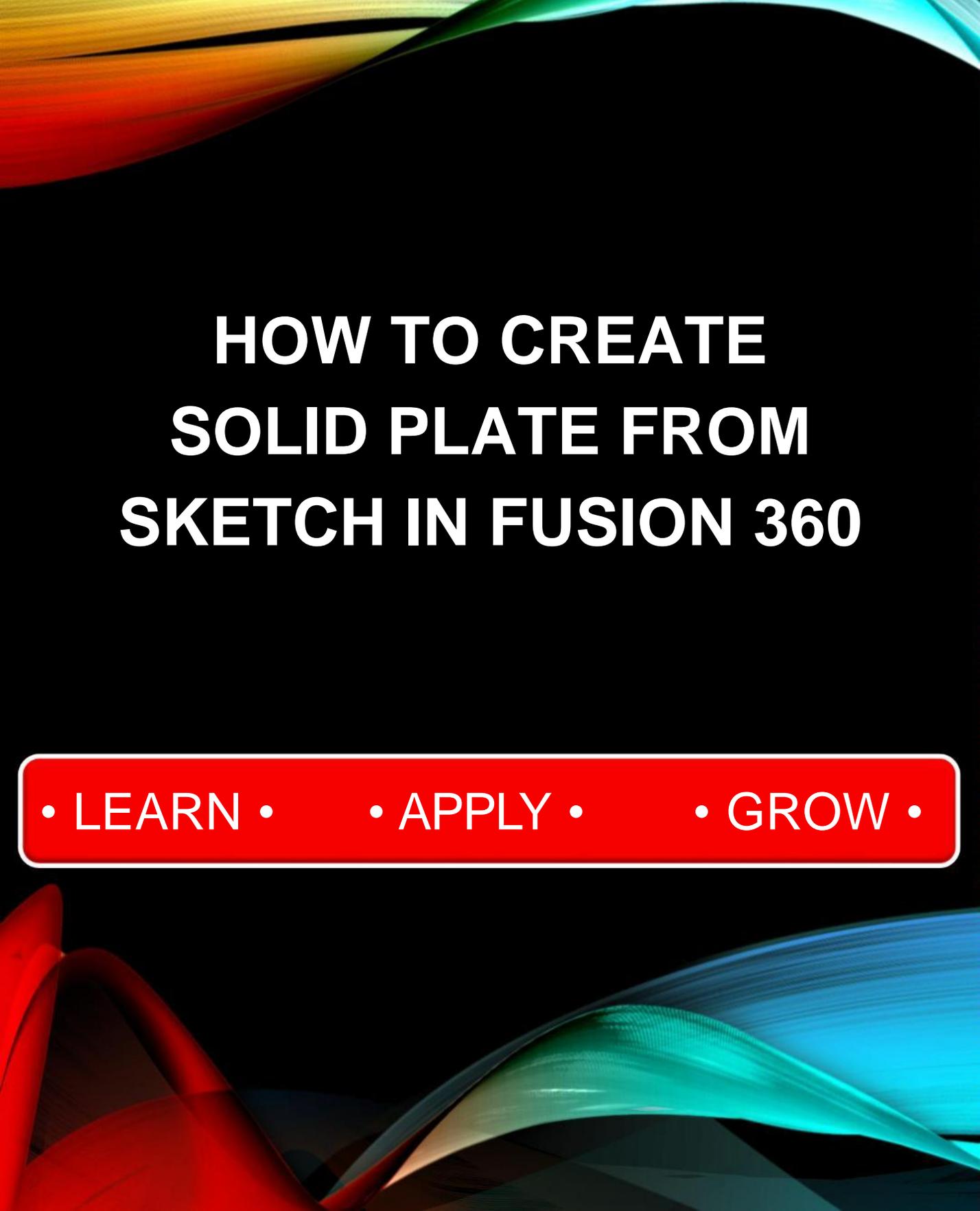
This blog has been created using a combination of artificial intelligence tools and human review to help deliver clear, structured, and up-to-date learning content.

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 CREATE SOLID PLATE FROM SKETCH IN FUSION 360

• LEARN • • APPLY • • GROW •

Introduction

Creating a solid plate from a sketch in Fusion 360 is a fundamental skill for anyone working in 3D modeling and CAD design. Whether you're designing a mechanical part, architectural component, or a custom project, mastering this process enables precise control over your models. This tutorial provides step-by-step guidance on how to transform your 2D sketch into a solid plate efficiently. We'll cover essential tools, best practices, common pitfalls, and practical examples. By following these instructions, you'll be able to produce accurate, manufacturable solid plates suitable for various engineering and design applications.

Understanding the Basics: Sketching to Solid Conversion in Fusion 360

Before diving into the step-by-step process, it's important to understand how Fusion 360 handles sketches and solids. Sketches are 2D profiles, and creating a solid from a sketch involves extruding, cutting, or forming these profiles into 3D objects. This process is central to parametric modeling, allowing modifications and iterations to be easily managed.

Fusion 360 offers powerful tools like Extrude, Loft, Revolve, and more, but for creating a solid plate, the **Extrude** function is typically the primary method. Let's explore how to use these tools effectively.

Step-by-Step Guide: How to Create a Solid Plate from a Sketch in Fusion 360

1. Starting a New Sketch

- Open Fusion 360 and start a new design.
- Select the appropriate plane or face to sketch on—commonly the XY, XZ, or YZ plane depending on your design orientation.
- Click on the **Create Sketch** button to initiate a new 2D sketch.

2. Drawing the Profile of Your Plate

- Use sketch tools such as **Rectangle**, **Circle**, or **Polygon** to draw the outline of the plate.
- Ensure your shape is closed; open profiles cannot be extruded into solids.
- Add dimensions using the **Sketch Dimension** tool to specify thickness, length, and width accurately.
- Use constraints (e.g., perpendicular, parallel, equal) to ensure your sketch is fully defined.

3. Finishing the Sketch

- Once your profile is complete and fully constrained, click **Finish Sketch**.
- Your 2D outline is now ready for extrusion.

4. Extruding the Sketch into a Solid Plate

- Activate the **Solid** tab in the toolbar.
- Select the **Extrude** tool.
- Click on the sketch profile to select it.
- Drag the arrow or input a specific value for the thickness of the plate.
- Choose the operation type: **New Body** (if creating a separate part), **Join** (to add to an existing body), or **Cut** (to remove material).
- Confirm the extrusion parameters and click **OK**.

5. Refining Your Solid Plate

- Use the **Fillet** or **Chamfer** tools to smooth edges or add features.
- Apply holes or cutouts for mounting or functional requirements by sketching on the plate surface and extruding cut features.
- Adjust dimensions as needed for precision.

Practical Example: Designing a Flat Mounting Plate

Suppose you want to create a flat mounting plate with holes for bolts:

- Sketch a rectangle 100 mm x 50 mm.
- Fully constrain the rectangle.
- Finish sketch.
- Extrude the rectangle to 5 mm thickness.
- Sketch circle profiles on the top face for bolt holes, say, 10 mm diameter.
- Extrude cut the circles through the entire thickness.

This simple example illustrates how to efficiently turn a 2D sketch into a practical flat plate.

Common Mistakes and How to Avoid Them

- **Unclosed profiles:** Ensure all sketch shapes are fully closed to allow proper extrusion.
- **Over-constrain sketches:** Avoid conflicting constraints; over-constraining can cause errors.
- **Not referencing the correct plane:** Double-check the sketch plane to ensure correct orientation.
- **Ignoring units:** Set and verify units before sketching to keep dimensions accurate.
- **Forgetting to finish the sketch:** Always finish your sketch before extruding.

Pro Tips for Creating Solid Plates in Fusion 360

- Use **Find and Fix Constraints** tool to troubleshoot sketch issues.
- Create a **Parameter** for your plate thickness to enable easy adjustments later.
- Leverage **Mirror** and **Pattern** features to replicate features quickly.

- When creating multiple cutouts or holes, sketch all features before extruding cuts to save time.
- Always double-check the direction of your extrude operation (upward or downward).

Best Practices for Solid Plate Design

- Maintain a clear sketch hierarchy: keep sketches simple and manageable.
- Use construction lines to set reference geometry.
- Regularly save versions of your design to prevent data loss.
- Double-check the size and position of all features before finalizing.
- Use the **Inspect** tool to analyze dimensions and integrity.

Comparing Fusion 360 Methods for Creating Solid Plates

Method	Use Case	Advantages	Limitations
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Extrude	Basic flat plate from a sketch	Simple, fast, widely used	Limited to extrusions, not complex shapes
Loft or Sweep	Unique shapes or complex profiles	Creates custom, flowing geometries	More advanced, requires multiple sketches
Revolve	Circular plates or symmetrical components	Precise round shapes	Only for revolved profiles

For creating solid plates with straightforward rectangular profiles, **extrude** remains the best choice due to its simplicity and flexibility.

Conclusion

Transforming a sketch into a solid plate in Fusion 360 is a foundational skill that empowers you to design precise, functional parts efficiently. By following structured steps—sketching accurately, fully defining your profile, and applying the **Extrude** tool—you can produce high-quality solid plates suitable for a wide array of applications. Incorporate best practices and avoid common mistakes to streamline your modeling process. With practice, these techniques will become second nature, significantly boosting your productivity and design quality.

FAQ

1. How do I create a rounded edge on my solid plate?

Ans: Use the **Fillet** tool to round edges after extruding the solid.

2. Can I add holes to a solid plate after extruding?

Ans: Yes, sketch circles on the plate surface and extrude cut through the thickness to create holes.

3. How do I change the thickness of my solid plate after creation?

Ans: Edit the original extrusion feature in the timeline or double-click the body and adjust the dimensions.

4. Is it possible to create a tapered or beveled edge on the plate?

Ans: Yes, apply a **Fillet** or use the **Chamfer** tool on edges for beveled or tapered effects.

5. What are common errors when creating a solid from a sketch in Fusion 360?

Ans: Common errors include unclosed profiles, over-constrained sketches, and referencing incorrect planes.

6. How can I ensure my sketch is fully constrained?

Ans: Use the **Sketch > Constraints** and **Dimensions** to add and verify constraints until the sketch is fully defined.

7. Can I create a complex hole pattern on my solid plate efficiently?

Ans: Yes, use **Pattern** features like rectangular or circular patterns after creating the initial hole to replicate efficiently.

About CADIN360

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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.

2026

Practice What You've Learned

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

500+ PRACTICE EXERCISES

• Sketching



2D Sketching

• 3D Modeling



3D Modeling

• Assembly



Assembly

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AUTODESK FUSION 360 ALL IN ONE WORKBOOK

500+ PRACTICE EXERCISES

2D Sketching • 3D Modeling • Assembly Drawings

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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.

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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

3D

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

- ✓ Books contains exercises of Sketching, 3D Modeling & Assembly.
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- ✓ Full Assembly STEP Files (.stp format) – Compatible with all major CAD software
- ✓ Get 200 3D Exercises in .f3d file format
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Thank You for Learning with Us!

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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)