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 EXTRUDE A SQUARE SKETCH IN FUSION 360

• LEARN • • APPLY • • GROW •

Introduction

Extruding a square sketch in Fusion 360 is a fundamental skill that forms the basis of many 3D modeling projects. Whether you're designing mechanical parts, architectural elements, or simple prototypes, mastering this process is essential. Fusion 360's intuitive interface, combined with its powerful extrude tools, allows users to create complex 3D models quickly and accurately. In this comprehensive guide, you'll learn how to extrude a square sketch step-by-step, along with tips for avoiding common mistakes and optimizing your workflow. If you're a beginner or looking to sharpen your skills, this post will provide clear instructions and practical insights to help you succeed.

How to extrude a square sketch in Fusion 360: Step-by-step process

Extruding a square sketch in Fusion 360 involves creating the initial 2D shape, then transforming it into a 3D object through extrusion. Here's a detailed walkthrough:

1. Start a new project and create a sketch

- Open Fusion 360 and click on the **File** menu to start a new design.
- Select the **Create Sketch** button from the toolbar.
- Choose the plane (XY, YZ, or ZX) where you want to draw the square.

2. Draw the square shape

- Use the **Rectangle** tool from the Sketch dropdown menu.
- Select the **Center Rectangle** or **Corner Rectangle** based on preference.
- Click on the sketch plane, then drag or input precise dimensions.

3. Define dimensions

- Use the **Sketch Dimension** tool (D) to assign specific side lengths.

- Enter the desired size for the sides, e.g., 50mm x 50mm.
- Make sure the sketch is fully constrained to avoid unintended modifications.

4. Finish the sketch

- Click **Finish Sketch** to exit sketch mode.
- Your square sketch is now ready for extrusion.

5. Select the sketch for extrusion

- Click on the face of the square or select the sketch in the Browser panel.
- Activate the **Solid** tab and click **Extrude**.

6. Set extrusion parameters

- In the Extrude dialog box:
- Enter the extrusion distance (e.g., 10mm for a cuboid).
- Select the **Direction** (One Side, Two Sides, or Symmetric).
- Choose whether to create a new body or join/create cut with existing geometry.

7. Complete the extrusion

- Click **OK** to generate the 3D shape.
- Your square sketch is now extruded into a solid block.

Practical examples of extruding squares in Fusion 360

Here are some real-world applications to put your knowledge into context:

- **Creating a simple box or housing**: Extrude a square sketch to form enclosures for electronics.
- **Designing mechanical components**: Generate shafts or brackets by extruding square profiles.
- **Rapid prototyping**: Quickly turn a 2D outline into a 3D prototype for initial testing or visualization.

Common mistakes and how to avoid them

Even experienced users encounter issues during extrusion. Here are some pitfalls and solutions:

- **Incorrect sketch constraints**: Unconstrained sketches can lead to unexpected extrusion results. Always fully constrain your drawings.
- **Wrong extrusion direction**: Make sure to select the correct direction—either one side, two sides, or symmetric.
- **Overlooking sketch plane**: Creating sketches on the wrong plane can cause misalignments. Confirm your sketch plane before drawing.
- **Not selecting the entire shape**: Ensure the entire square profile is selected before extrusion; otherwise, only a portion will extrude.

Pro tips for efficient extruding in Fusion 360

- Use **Create Components** if working on complex assemblies, facilitating better control.
- Save your work frequently to prevent data loss.
- Use **Dimension Constraints** for precise control over your sizes.
- Experiment with **Change Parameters** for parametric modeling—adjust dimensions post-extrusion easily.
- Apply **Fillets** or **Chamfers** after extrusion to refine your edges.

Comparing extrusion techniques: Freeform vs. Direct Extrude

While direct extrusion is straightforward for simple shapes like squares, Fusion 360 also offers freeform tools for more complex geometries:

Technique	Best Use Case	Pros	Cons
Direct Extrude	Simple shapes, perfect cubes	Fast, precise, easy to learn	Limited to straight profiles
Freeform	Organic, complex shapes	Flexible, sculpting-like	Steeper learning curve

For most beginner and intermediate projects, the **direct extrusion** method suffices, especially when extruding basic shapes like squares.

Conclusion

Mastering how to extrude a square sketch in Fusion 360 unlocks endless design possibilities, from creating simple objects to complex assemblies. By following the step-by-step instructions, avoiding common pitfalls, and leveraging pro tips, you can achieve precise and efficient results. Whether you're prototyping, manufacturing, or learning CAD fundamentals, extrusion is a core function worthy of mastery. Practice regularly, experiment with different parameters, and you'll quickly become proficient in transforming your 2D sketches into functional 3D models.

FAQ

1. How do I ensure my square sketch is fully constrained in Fusion 360?

Ans : Use the dimension tool to specify all side lengths and ensure the shape is fully constrained with no unresolved markers.

2. Can I extrude multiple sketches at once in Fusion 360?

Ans : Yes, you can select multiple profiles and extrude them simultaneously by holding down the Ctrl key or using the selection box.

3. What is the typical extrusion distance for creating a cube?

Ans : Set the extrusion distance to match your side length for a perfect cube, e.g., 50mm if your square's sides are 50mm.

4. How do I create a hollow square or frame shape in Fusion 360?

Ans : Draw a larger square and an inner smaller square within your sketch, then extrude the outer shape and cut the inner profile using a cut operation.

5. Is it possible to extrude a square to a negative value for cut operations?

Ans : Yes, select the sketch profile and set a negative extrusion distance to cut into existing geometry.

6. How can I modify an extruded square after creating it?

Ans : Edit the sketch to change dimensions or use the **Press Pull** tool to adjust the extrusion height directly.

7. Can I animate or simulate the extrusion process in Fusion 360?

Ans : Fusion 360 does not support animation of extrusion directly, but you can create a time-lapse or simulate parameters through functional modeling techniques.

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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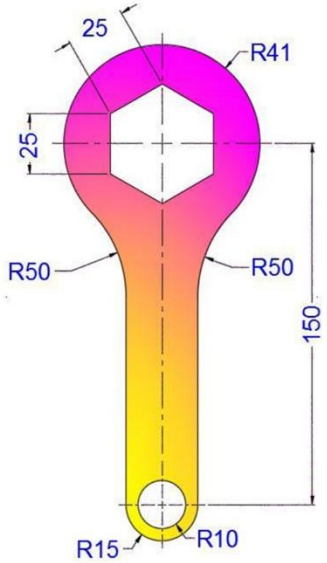
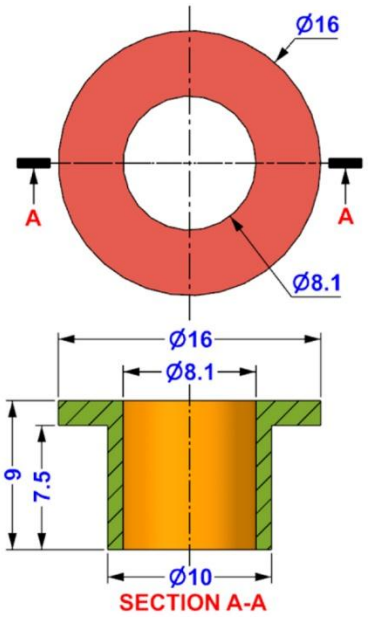
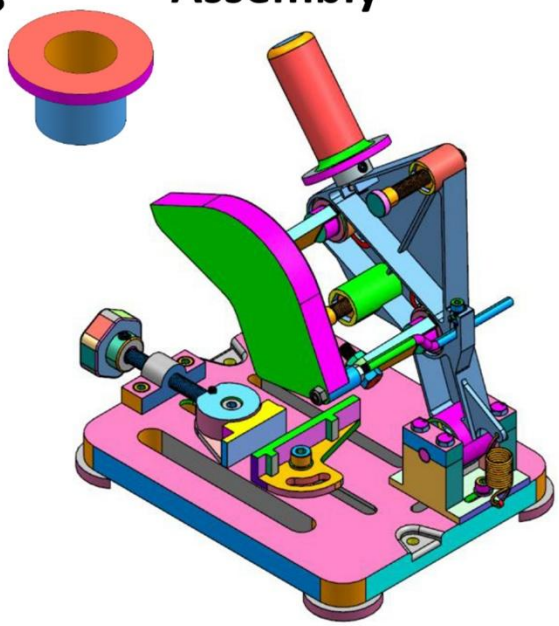
- Assembly



2D Sketching



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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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:

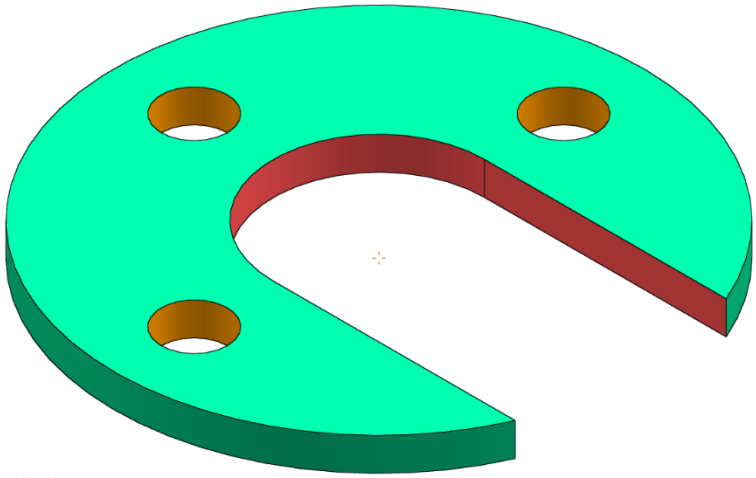
- Complete the exercises in order, or jump to any skill level you prefer.
- All dimensions are in millimeters.
- Where dimensions are missing, apply logic or practice estimation.
- This book is ideal for both students and professionals preparing for industry design work.

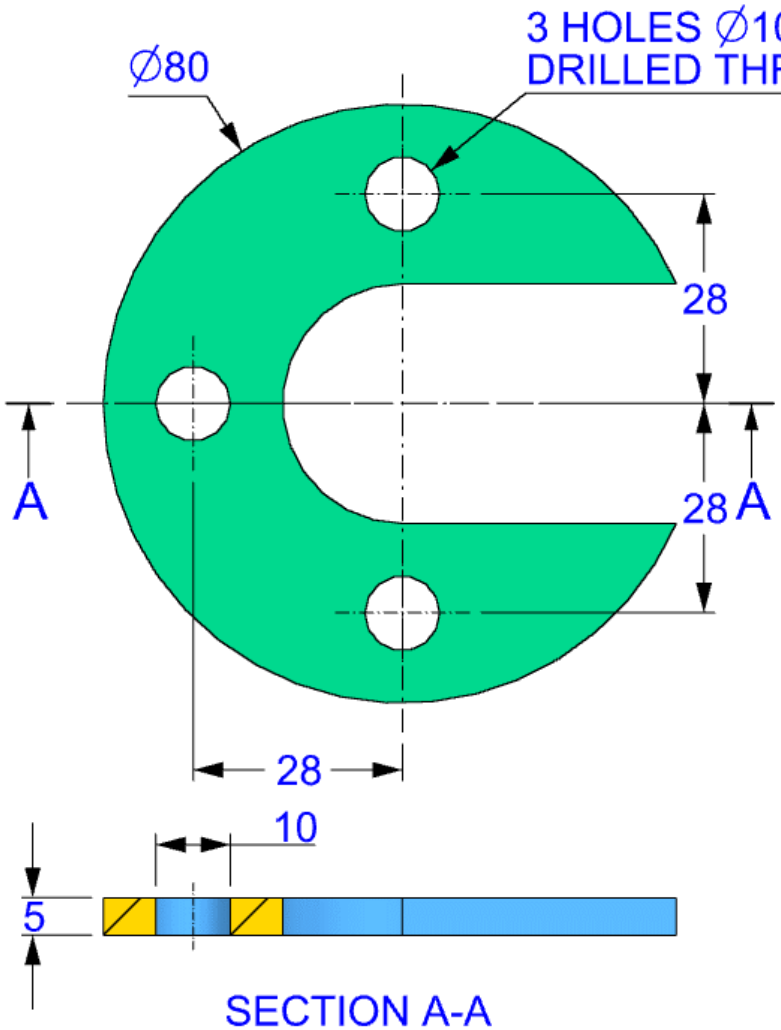
Note:

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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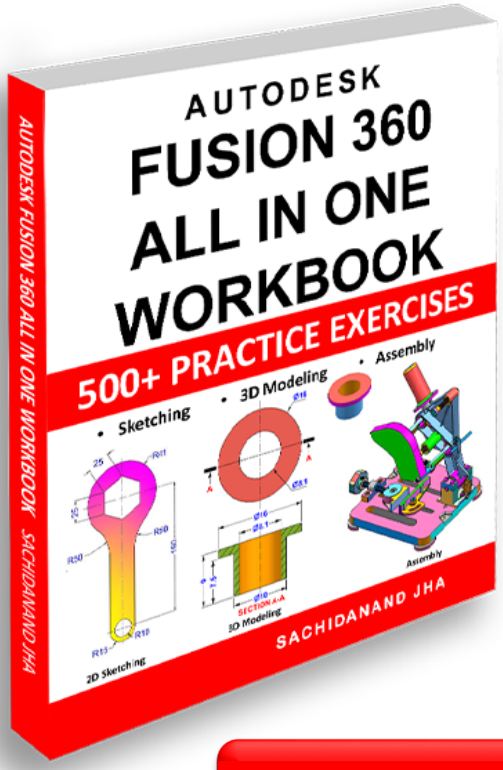

What's Included in the FUSION 360 ALL IN ONE WORKBOOK?

- ✓ 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)