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 EXTRUDE A CIRCULAR SKETCH IN FUSION 360

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Introduction

Extruding a circular sketch in Fusion 360 is a fundamental process that allows designers and engineers to create three-dimensional objects from 2D profiles. Whether you're building a mechanical part, a decorative component, or a simple cylinder, understanding how to properly extrude a circular sketch is essential in Fusion 360. In this comprehensive guide, we'll walk through each step involved in extruding a circular sketch, discuss best practices, common mistakes to avoid, and share tips to enhance your modeling workflow. By mastering this technique, you'll be able to create precise, professional parts efficiently.

How to Extrude a Circular Sketch in Fusion 360

Extruding a circular sketch involves creating a 2D circle, then extending it in 3D space to give it volume. This straightforward process is central to many modeling projects, from basic shapes to complex assemblies. Below, you'll find detailed instructions suitable for beginners and seasoned users alike.

Step-by-step instructions for extruding a circular sketch

1. Create a new sketch on the desired plane

- Open Fusion 360.
- Select **Create Sketch** from the toolbar.
- Choose the plane where you'd like to draw the circle (XY, YZ, XZ, or any custom plane).

2. Draw the circle

- Select the **Center Diameter Circle** tool from the Sketch dropdown.
- Click on the sketch plane to set the circle's center point.
- Drag outward or input a specific diameter value in the dialog box.

Tip: Use constraints like *Diameter or Radius* to set exact dimensions for precision.

3. Finish the sketch

- Click **Finish Sketch** in the toolbar once your circle is complete.
- Your 2D circular profile is now ready for extrusion.

4. Activate the Extrude command

- Select the **Solid** tab, then click **Extrude**.
- Alternatively, right-click your sketch in the Browser panel and select **Extrude**.

5. Select the circular profile

- Click on the circle in your sketch to highlight it.
- If multiple profiles exist, ensure only the intended circle is selected.

6. Define extrude parameters

- Enter the desired distance for the extrusion (e.g., 20 mm).
- Choose the direction of extrusion:
- **One Side:** Extends in one direction.
- **Symmetric:** Extends equally in both directions.
- **Two Sides:** Extends in both directions for specific thicknesses.

7. Confirm and complete the extrusion

- Click **OK**.
- Your circular sketch is now converted into a 3D solid cylinder or shape based on your parameters.

Practical example: Creating a cylindrical peg

To illustrate, imagine you'd like to make a cylindrical peg:

- Draw a circle with a diameter of 10 mm.
- Extrude the circle 50 mm in one direction.
- Finish with a chamfer or fillet if necessary for real-world application.

This example showcases how straightforward it is to produce functional parts using extrusion.

Common Mistakes When Extruding Circular Sketches

Understanding what errors to avoid can save you time and frustration. Here are some typical pitfalls:

1. Forgetting to finish the sketch

- **Mistake:** Attempting to extrude before completing the sketch.
- **Solution:** Always click **Finish Sketch** after drawing before extruding.

2. Not selecting the correct profile

- **Mistake:** Accidentally selecting overlapping or multiple profiles.
- **Solution:** Carefully click on your profile or use selection filters to avoid unwanted geometry.

3. Ignoring constraints for dimension accuracy

- **Mistake:** Using freehand circles without constraints.
- **Solution:** Apply diameter or radius constraints early for precise control.

4. Incorrect direction or distance

- **Mistake:** Extruding in the wrong direction or setting an inappropriate distance.
- **Solution:** Double-check the direction options and input accurate measurements.

5. Overlooking the need for additional features

- **Mistake:** Forgetting to add fillets or chamfers after extrusion.
- **Solution:** Use features like **Fillet** or **Chamfer** to refine your part post-extrusion.

Pro Tips for Better Circular Extrusions

Implementing these best practices can elevate your modeling efficiency:

- **Use the 'Direction' options** in extrude to control how material extends.
- **Apply parameters for repeatability**: Use parameters and formulas for dimensions, making modifications easier.
- **Leverage symmetry**: When creating symmetrical objects, select the **Symmetric** option for uniform extrusion.
- **Combine extrusions**: For complex shapes, consider combining multiple extrusions with different profiles.
- **Use the 'Extent' feature**: For advanced extrusions, options like 'To Object' or 'From Object' can help create precise cuts or holes.

Creating Complex Shapes from Circular Extrusions

A simple circular extrusion can be transformed into complex features:

- **Cutouts:** Create smaller circles and extrude cut to form holes.
- **Ribs and supports:** Extrude multiple profiles to build structural elements.
- **Tapered shapes:** Use the **Taper Angle** option to create cones or tapered cylinders.

Fusion 360 offers extensive tools to refine your circular extrusion, making it a versatile stage in your design process.

Comparing Extrude Options in Fusion 360

Fusion 365 provides various extrusion methods that suit different needs:

Method	Description	Usage Example
One Side	Extrudes in one direction from sketch plane	Creating solid cylinders
Symmetric	Extrudes equally in both directions	Thin-walled cylinders or tubes
Two Sides	Extends in both directions with different distances	Brackets with back-to-back features

Choosing the right method ensures your design process is efficient and results are precise.

Conclusion

Mastering how to extrude a circular sketch in Fusion 360 is essential for anyone serious about 3D modeling. This process enables you to turn simple 2D sketches into complex 3D objects with accuracy and ease. Whether creating basic cylinders, intricate mechanical components, or decorative elements, understanding the steps, avoiding common errors, and applying expert tips will streamline your workflow and improve your designs. With practice, extrusion becomes a quick, intuitive tool in your CAD toolkit, opening the door to more advanced modeling techniques and innovative projects.

FAQ

1. How do I create a hole in a solid cylinder in Fusion 360?

Ans: Draw a smaller circle on the surface of the cylinder's face, then extrude cut through the material.

2. Can I extrude a circle in multiple directions at once?

Ans: Yes, by selecting the **Two Sides** extrusion option and defining distances for both directions.

3. How do I create a tapered circular extrusion?

Ans: During the extrusion, enable the **Taper Angle** option and specify the desired angle.

4. What is the best way to create a hollow cylinder?

Ans: Draw two concentric circles with different diameters, extrude the larger one, then extrude cut the smaller circle inside.

5. How do I scale an existing circular extrusion in Fusion 360?

Ans: Use the **Scale** feature in the **Modify** menu to resize your solid proportionally.

6. Can I edit the dimensions of my circular extrusion later?

Ans: Yes, simply double-click the sketch or feature in the timeline and modify the dimensions.

7. How do I ensure my extrusion is precise and matches engineering drawings?

Ans: Use constraints and dimension inputs during sketching, and double-check measurements before extruding.

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.

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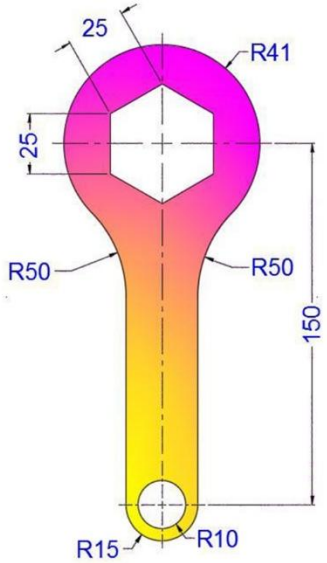
- Assembly



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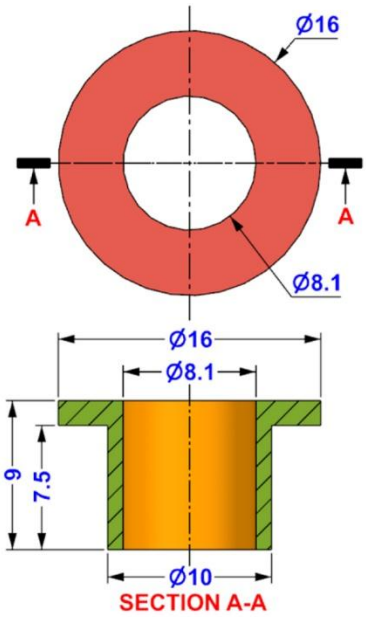
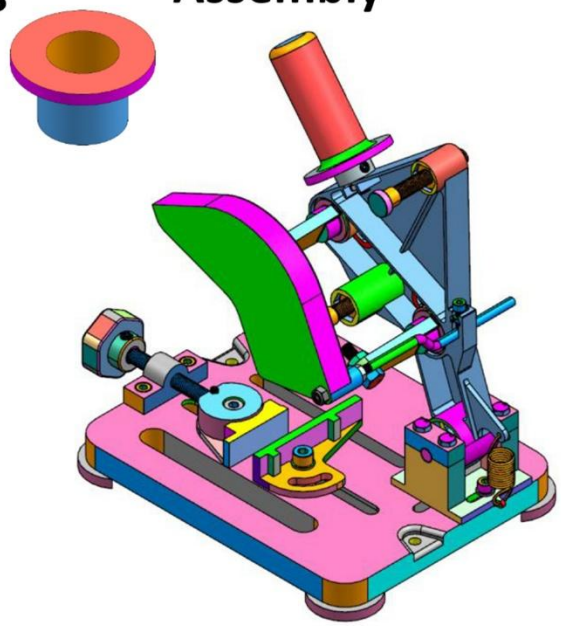


3D Modeling



Assembly

SACHIDANAND JHA

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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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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.

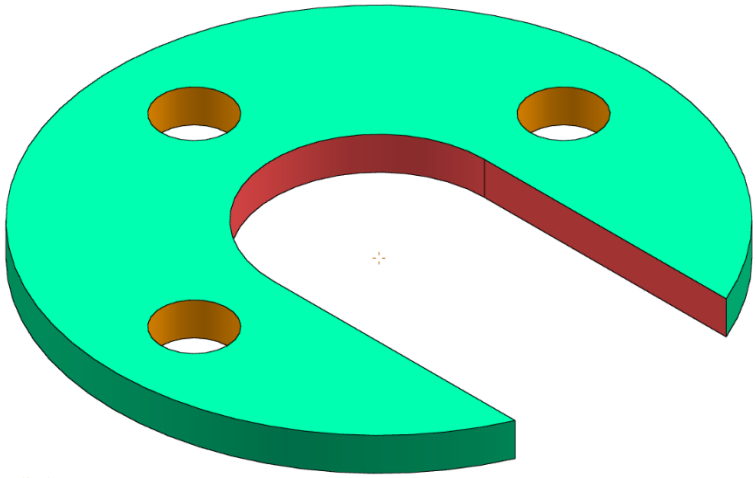
Tips for Best Use:

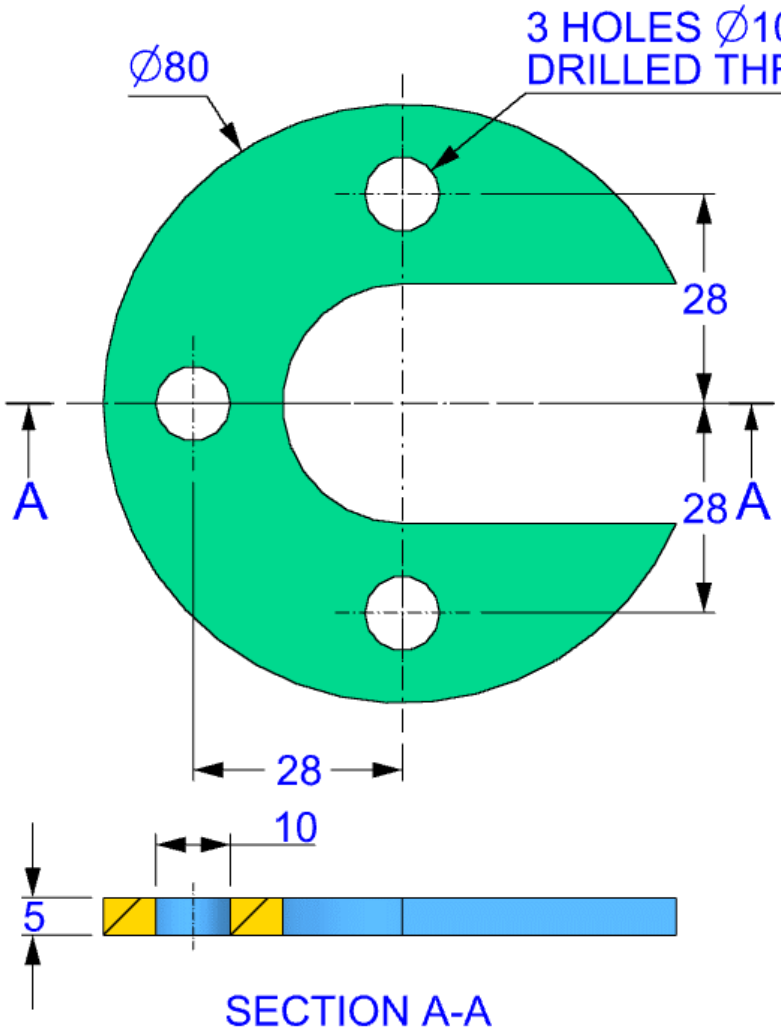
- Complete the exercises in order, or jump to any skill level you prefer.
- All dimensions are in millimeters.
- Where dimensions are missing, apply logic or practice estimation.
- This book is ideal for both students and professionals preparing for industry design work.

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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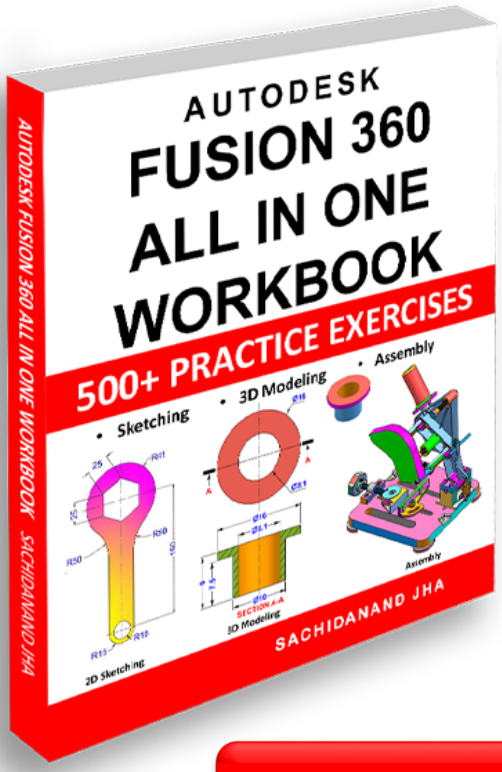

What's Included in the FUSION 360 ALL IN ONE WORKBOOK?

- ✓ 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
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We're honored to be a part of your journey.
Happy Designing!
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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)