

AUTODESK FUSION 360

2026

# BLOG

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**cadin360°**  
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

# DIFFERENCE BETWEEN PRESS PULL AND EXTRUDE IN FUSION 360

• LEARN •      • APPLY •      • GROW •

# Introduction

When working in Fusion 360, understanding the fundamental difference between the various modeling tools is crucial for efficient and accurate design. Among these tools, **Press Pull** and **Extrude** are two widely used features that serve distinct purposes. While both modify 3D geometry, they differ significantly in their application, flexibility, and outcomes. This comprehensive guide explores the key differences between press pull and extrude in Fusion 360, providing step-by-step instructions, practical examples, and best practices to help you choose the right tool for your design needs. Whether you're a beginner or an experienced CAD user, mastering these commands will elevate your modeling skills.

## Understanding Fusion 360's Modeling Tools: Press Pull vs. Extrude

At a glance, *press pull* and *extrude* might seem similar because both involve modifying solid bodies by adding or removing material. However, they are fundamentally different tools designed for specific scenarios.

### What is the Extrude Tool?

Extrude is one of the core features in Fusion 360's solid modeling environment. It allows you to create a 3D feature from a 2D sketch profile by extending it in a specific direction. Extruding can be used to create new bodies, cut features, or join components, depending on its settings.

### What is the Press Pull Tool?

Press pull is a more flexible and interactive tool that allows you to modify existing geometry directly. It enables you to push or pull faces, edges, or regions of a body without necessarily starting from a sketch. It's ideal for quick adjustments, complex shape modifications, or localized edits.

## Step-by-Step: Using the Extrude Command in Fusion 360

The extrude tool is typically used during the initial modeling phase or for creating features from sketches.

## 1. Creating a Sketch

- Select a plane or face where you want to begin your extrusion.
- Use sketch tools such as rectangle, circle, or polygon to draw your profile.

## 2. Initiate the Extrude

- Click on the *Create* menu, then select *Extrude* or press the keyboard shortcut **E**.
- Select the sketch profile you wish to extrude; it will highlight when clicked.

## 3. Set the Extrude Parameters

- Drag the arrow to visually size the extrusion or input a precise distance.
- Choose the direction: one side, two sides, or symmetric.
- Decide whether to create a new body, join to existing, or cut from existing geometry.

## 4. Complete the Operation

- Click **OK** to finalize the extrusion.
- Your shape is now a 3D solid that can be further modified or used as a base for additional features.

## Practical Example

Suppose you want to create a rectangular box. You sketch a rectangle, then extrude it upward to form a solid block.

## Common Mistakes to Avoid

- Forgetting to select the correct sketch profile.

- Not setting the correct operation type (join, cut, or new body).
- Exceeding or undercutting dimensions due to accidental input errors.

## Step-by-Step: Using the Press Pull Tool in Fusion 360

The press pull tool offers a more intuitive approach to modifying existing geometry, often used for quick adjustments or complex surface edits.

### 1. Select the Face or Region

- Click on the face or region you wish to modify.
- You can select multiple faces or edges for complex adjustments.

### 2. Activate Press Pull

- Right-click the selection and choose **Press Pull** from the context menu.
- Alternatively, click the *Modify* menu and select **Press Pull** or press **Q**.

### 3. Push or Pull the Geometry

- Drag the arrow or face to extend or retract the selected area.
- Move it interactively, or input a precise distance in the dialog box.

### 4. Adjust the Options

- Decide whether to create a new feature or cut into existing geometry.
- You can also select to "Keep Faces" or "Remove Material," depending on the goal.

### 5. Confirm and Finish

- Click **OK** to apply the modifications.
- The geometry updates immediately, allowing further adjustments if needed.

## Practical Example

Suppose you have a machined plate with a hole, and you want to slightly increase the hole's diameter. Using press pull on the hole perimeter can quickly reshape it without needing to redo the sketch.

## Common Mistakes to Avoid

- Selecting the wrong face or region, leading to unintended modifications.
- Applying press pull without considering the surrounding geometry.
- Forgetting to specify whether it's a cut or join operation.

## Comparing Press Pull and Extrude: Key Differences

<b>Feature</b>	<b>Extrude</b>	<b>Press Pull</b>
Primary Use	Creating 3D features from sketches	Modifying existing geometry directly

Starting Point	Requires a 2D sketch profile	Operates directly on faces, edges, or regions
Flexibility	Good for creating new shapes and features	Excellent for quick edits and adjustments
Interaction	Less interactive, based on specific parameters	Highly interactive with visual feedback
Geometries Created	Adds new bodies or features	Changes existing geometry in place

Ideal Scenarios	Building new parts or features from scratch	Fine-tuning, making minor adjustments, or complex shape edits
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## Best Practices and Tips

- Use extrude when creating a new feature from a sketch or to add material.
- Use press pull for quick modifications or when fine-tuning existing geometry.
- Always double-check selected regions before confirming a press pull operation.
- Combine both tools for complex assemblies—extrude to create, press pull to refine.
- Save versions or duplicate bodies before making major modifications for easy rollback.

## Common Mistakes and How to Avoid Them

- **Mistake:** Using extrude when a quick, localized adjustment is needed.

**Pro tip:** Use press pull for faster edits to existing geometry.

- **Mistake:** Selecting the wrong face or sketch profile.

**Pro tip:** Use highlight and preview features to confirm your selection.

- **Mistake:** Overlooking the operation type (join, cut, or new body).

**Pro tip:** Be deliberate about the operation type to prevent errors.

## Practical Tips for Beginners

- Practice sketching accurately, as this benefits extrude operations.
- Experiment with press pull on various geometries to understand how it manipulates surfaces.
- Use the timeline to revisit and modify features after applying press pull.
- Combine extrude and press pull with other tools like fillet, chamfer, or shell to create complex designs.

## Conclusion

Understanding the fundamental difference between press pull and extrude in Fusion 360 empowers you to model more efficiently and accurately. **Extrude** is best suited for building new features from scratch using sketches, offering precision and control. **Press pull** excels in editing and fine-tuning existing geometry quickly and interactively. Mastery of both tools, along with knowing when to use each, is essential for producing high-quality 3D models and streamlining your CAD workflow.

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## FAQ

### 1. What is the main difference between press pull and extrude in Fusion 360?

**Ans:** Extrude creates new 3D features from sketches, while press pull modifies existing geometry directly by pushing or pulling faces or regions.

### 2. Can I use press pull to create a new feature from scratch?

**Ans:** No, press pull is designed to edit existing geometry; for creating new features, the extrude tool is more appropriate.

### 3. Is press pull suitable for complex surface modifications?

**Ans:** Yes, press pull is ideal for quick, complex adjustments on surfaces or regions without needing new sketches.

#### 4. How do I decide whether to use extrude or press pull?

**Ans:** Use extrude when creating new features from sketches, and press pull for quick edits or adjustments to existing geometry.

#### 5. Can I combine extrude and press pull in my workflow?

**Ans:** Absolutely, combining both tools allows for precise modeling and quick modifications, streamlining your CAD process.

#### 6. What are some common mistakes when using extrude?

**Ans:** Common mistakes include selecting the wrong sketch profile, incorrect operation type, or miscalculating the extrusion distance.

#### 7. What are best practices for using press pull effectively?

**Ans:** Always preview your selection, choose the correct operation (cut or join), and make incremental adjustments for best results.

# 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.

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

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### • Sketching



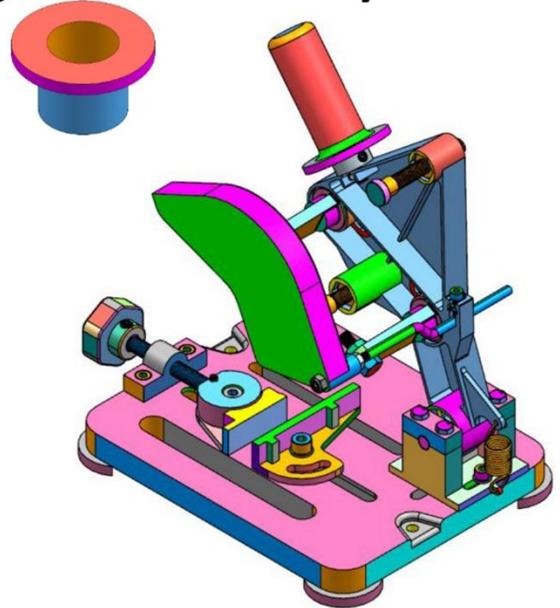
2D Sketching

### • 3D Modeling



3D Modeling

### • Assembly



Assembly

SACHIDANAND JHA

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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](mailto:cadin360@gmail.com)

If you think you've found a technical error in this book, please visit <https://cadin360.com/contact-us/>.

Customer feedback is critical to our efforts at CADIN360.

Best regards,

Sachidanand Jha  
Founder & CEO, CADIN360



# **AUTODESK FUSION 360 ALL IN ONE WORKBOOK**

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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.

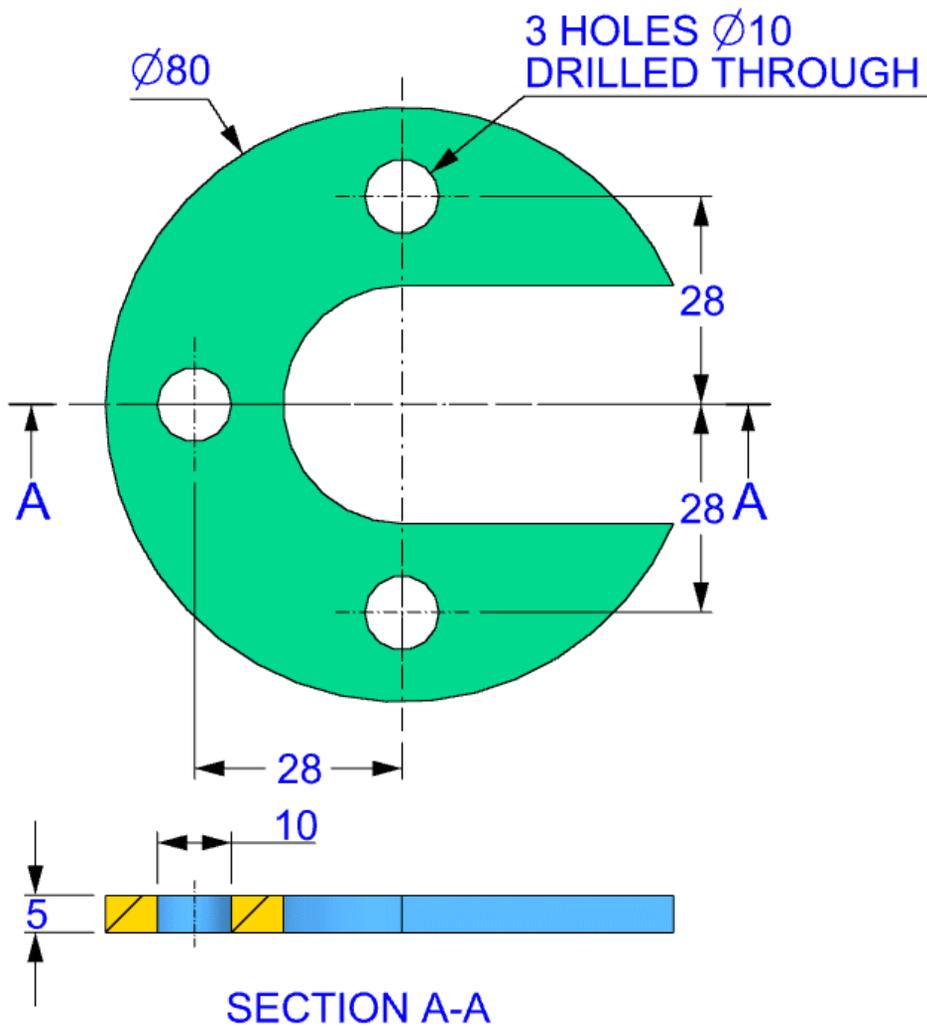
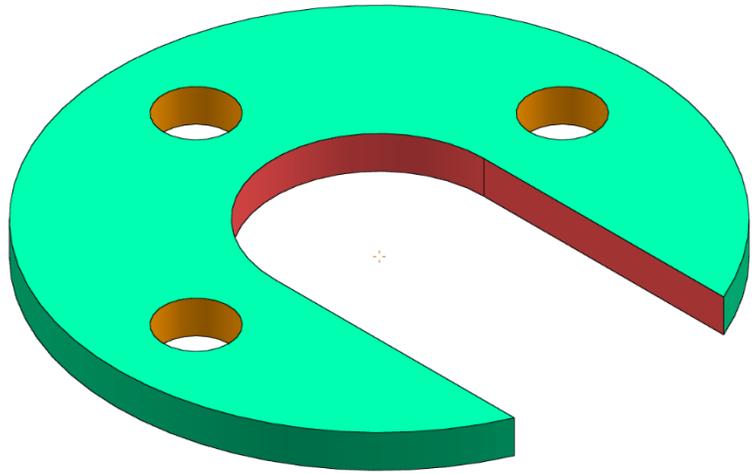
### **Note:**

This book is available in multiple formats – **Black & White**, **Standard Color**, and **Premium Color** editions.

Happy learning!  
– Team CADIN360

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## EXERCISE-01



# Get The Complete Practice Sample

You downloaded a single Exercise PDF

The complete practice sample for this software includes multiple exercises and is not available inside this PDF..

## What you will receive

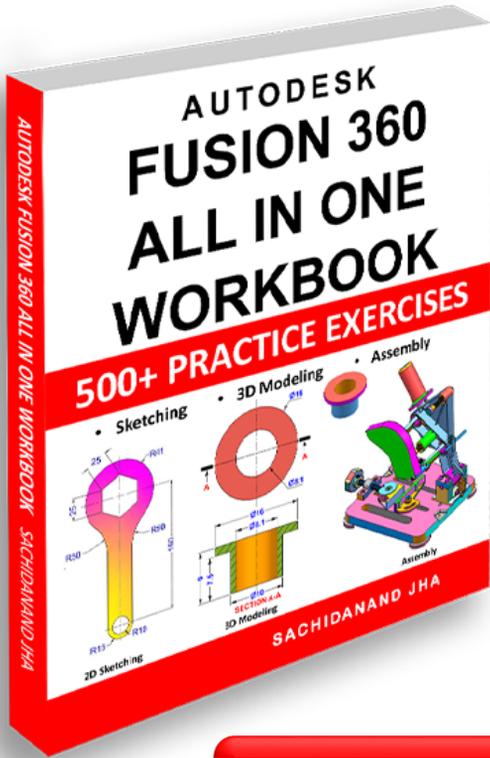
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# END OF SAMPLE



## 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
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## 🚀 Keep Practicing. Keep Designing.

Whether you're a beginner or a pro, **practice is the key** to mastering any CAD software.

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 3<sup>rd</sup> Angle projection
- All dimensions are in mm(metric system)