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 INCREASE HEIGHT OF SOLID IN FUSION 360

• LEARN • • APPLY • • GROW •

Introduction

Increasing the height of a solid body in Fusion 360 is a fundamental skill for many design projects. Whether you're creating prototypes, mechanical parts, or structural components, understanding how to efficiently modify solid height is crucial. Fusion 360's versatility with direct modelling and parametric features makes it easier to adjust solid bodies with precision and control. In this guide, we'll walk through the best methods to increase the height of a solid in Fusion 360, including practical steps, common mistakes to avoid, and expert tips to optimize your workflow.

How to Increase Height of a Solid in Fusion 360

Adjusting the height of a solid in Fusion 360 involves different techniques depending on your design intent and the nature of the model. Below, we detail the most effective methods to achieve this.

1. Using the Extrude Tool to Extend a Solid

The Extrude tool is one of the core features in Fusion 360 used to modify the height of solid bodies.

- Open Your Design: Launch Fusion 360 and open the model you want to modify.
- Select the Body or Sketch:
- If your solid is based on a specific sketch, select that sketch in the Browser.
- Alternatively, select the face or body directly in the canvas.
- Activate the Extrude Tool:
- Go to the 'Solid' tab.
- Click on 'Create' > 'Extrude' or press the shortcut 'E'.
- Adjust the Extrude Distance:
- In the dialog box, enter the amount you want to increase the height.
- Choose whether to extend the existing solid or cut into it.

- Define the Direction:
- Choose 'Symmetric' if extending both sides equally.
- Set the direction as 'One Side' to extend in a specific direction.
- Complete the Operation:
- Confirm by clicking OK.
- Your solid body now has increased height.

Pro Tip: Use the 'Direction' options wisely to control whether you're adding height to the top, bottom, or both sides.

2. Using the Press Pull Tool to Adjust Height

The Press Pull tool allows you to modify the height directly by selecting specific faces or regions.

- Select the Face:
- Click the top face of your solid that you want to extend.
- Activate Press Pull:
- Under the 'Modify' menu, select 'Press Pull' or press 'Q'.
- Drag or Input Distance:
- Drag the face upward to visually increase height.
- Or, enter an exact numerical value for precision.
- Confirm:
- Click OK to complete the operation.

This method is especially useful for quick adjustments or when working on irregular geometries.

3. Creating and Using Components for Modular Height Adjustment

For complex models or repeatable parts, Creating components can make height modifications more manageable.

- Create a New Component:
- Right-click the top-level browser and select 'New Component'.
- Design or select the part you want to modify.
- Use Parameters:
- Define parameters for height, such as 'HeightIncrement'.
- This makes controlling and changing height easier across iterations.
- Apply Parametric Changes:
- Change the parameter value.
- Fusion 360 automatically updates the component's height accordingly.

This approach is highly efficient for projects requiring multiple adjustments or variations.

4. Employing the Scale Tool for Uniform Height Increase

The Scale tool can resize your entire solid uniformly or along specific axes.

- Select the Solid Body:
- Click on the body in the Browser or canvas.
- Activate the Scale Tool:
- Go to 'Modify' > 'Scale'.
- Choose the Type:
- Select 'Non-Uniform' scale.
- Adjust the scale factor along the Z-axis (height).
- Input Scaling Factor:
- Enter a value greater than 1 to increase height.
- Confirm:

- Click OK.

Note: Be cautious with scaling, as it alters the entire model proportionally, which may not always be desirable.

5. Using the Loft or Sweep Features for Custom Height Adjustments

For more complex height modifications, especially in curved or irregular bodies, Loft or Sweep features are powerful options.

- Create Sketch Profiles:
- Sketch the initial and final profiles with desired heights.
- Use Loft:
- Select 'Create' > 'Loft'.
- Connect the profiles to form a solid with the new height.
- Use Sweep:
- Create a profile and path.
- Sweep the profile along the path to build a custom height.

This method offers high precision for complex shape modifications.

Practical Examples and Applications

Let's explore real-world scenarios to solidify these methods.

- **Example 1:** Extending a Base Plate
- Use the Extrude tool to add height uniformly to the base of a mechanical part.
- **Example 2:** Adjusting the Height of a Themed Component
- Employ Press Pull for quick height modifications on irregular surfaces.
- **Example 3:** Creating Parametric Models for Prototypes

- Set up parameters for height in the component to enable easy adjustments later.

Applying these techniques contextually ensures swift, efficient modifications tailored to your project needs.

Common Mistakes to Avoid

- Over-extruding without considering design constraints.
- Not updating parameters when models depend on variable heights.
- Scaling entire bodies unintentionally, distorting the overall design.
- Ignoring the implications of changing dimensions on assemblies.
- Forgetting to turn off constraints that might restrict height modifications.

Being mindful of these pitfalls helps maintain model integrity.

Pro Tips and Best Practices

- Always work with parametric dimensions when precise control is needed.
- Use named parameters for easy updates and version control.
- Combine multiple techniques for complex modifications.
- Regularly save versions before significant changes.
- Use the ‘Inspect’ tool to verify dimensions after modifications.

Following these best practices enhances your efficiency and accuracy.

Comparing Fusion 360 Techniques to Other CAD Software

Method	Fusion 360	SolidWorks	AutoCAD
Extrude	Yes	Yes	Yes
Press Pull	Yes	No	No
Scale	Yes	Yes	Yes
Loft/Sweep	Yes	Yes	Yes
Parametric Modeling	Yes	Yes	Limited

Fusion 360 offers a versatile combination of tools for height adjustments, often more integrated than traditional CAD software.

Conclusion

Increasing the height of a solid body in Fusion 360 is straightforward once you understand the right techniques and tools. Whether you prefer the classic Extrude method, the quick Press Pull, or parametric adjustments for flexible design variations, Fusion 360 has the right solution for your needs. Mastering these methods enhances your modeling efficiency and ensures your designs meet precise specifications. By practicing these steps and avoiding common mistakes, you can confidently modify solid heights to fit your project requirements.

FAQ

1. How do I increase the height of a solid body in Fusion 360 without affecting other parts?

Ans: Use the Extrude tool to extend only the selected face or body, ensuring other parts remain unaffected.

2. Can I change the height of a component later in Fusion 360?

Ans: Yes, by linking the component's height to a parameter, you can easily modify it later.

3. What's the best way to increase height without distorting the entire model?

Ans: Use the press pull or extrude tools directly on the specific face or feature you want to modify.

4. How do I uniformly scale a solid to increase its height in Fusion 360?

Ans: Use the Scale tool, select the body, and adjust the scale factor along the Z-axis for a uniform increase.

5. Is it possible to increase height after creating complex features or sketches?

Ans: Yes, by editing the original sketches or using lofts and sweeps to add the desired height with precision.

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.

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FUSION 360 ALL IN ONE WORKBOOK

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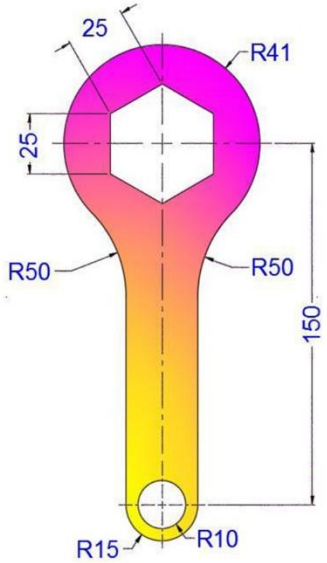
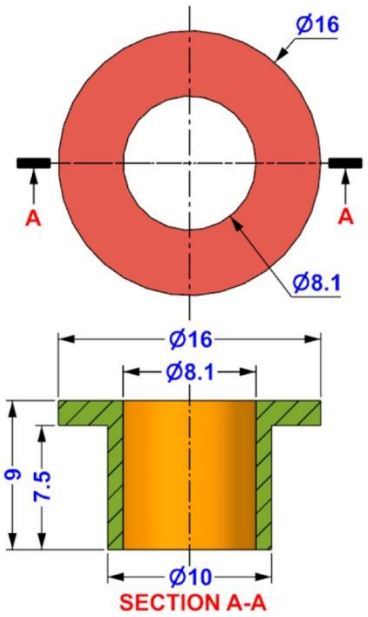
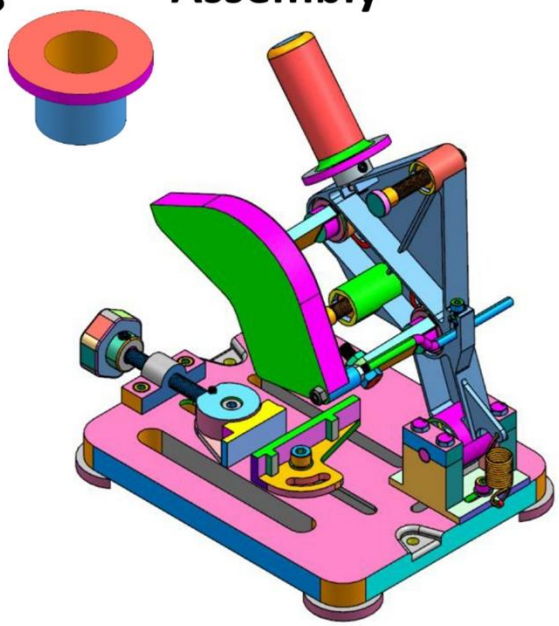
- 3D Modeling
- Assembly



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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Customer feedback is critical to our efforts at CADIN360.

Best regards,

Sachidanand Jha
Founder & CEO, CADIN360

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Published by 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:

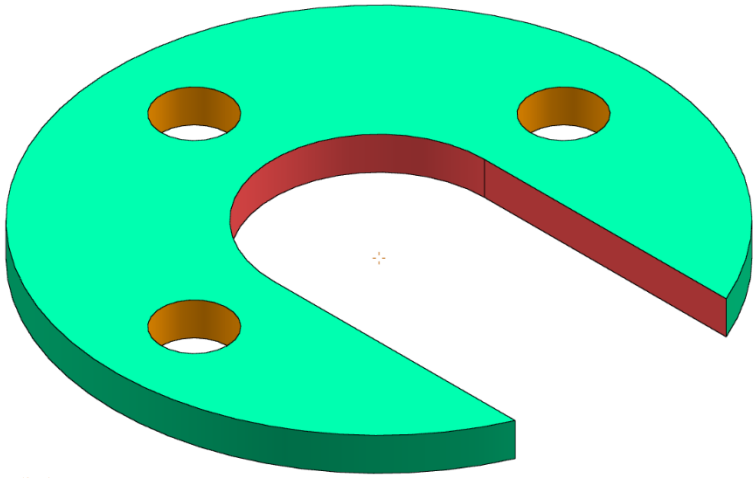
- Complete the exercises in order, or jump to any skill level you prefer.
- All dimensions are in millimeters.
- Where dimensions are missing, apply logic or practice estimation.
- This book is ideal for both students and professionals preparing for industry design work.

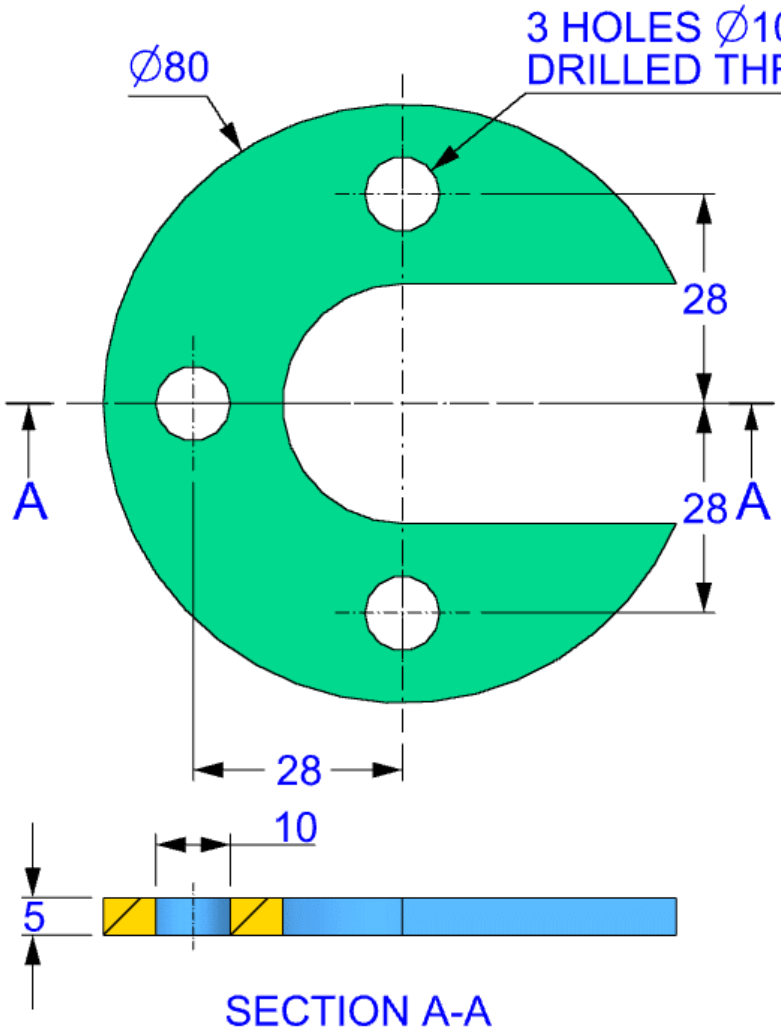
Note:

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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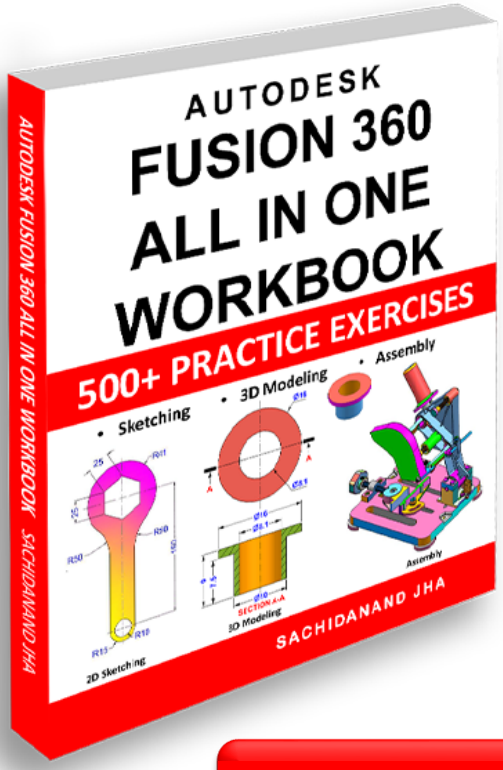

What's Included in the FUSION 360 ALL IN ONE WORKBOOK?

- ✓ 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!
– 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)