

AUTODESK FUSION 360

2026

# BLOG

# 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



# WHY SOLID TOOLS ARE GREYED OUT IN FUSION 360

• LEARN •      • APPLY •      • GROW •

# Introduction

Solid tools are fundamental features within Fusion 360 that allow users to create and modify model geometry efficiently. However, many users encounter a common issue: their solid tools suddenly become greyed out or unavailable. This can be frustrating, especially when you're in the midst of a project. Understanding why solid tools are greyed out in Fusion 360 is key to resolving the problem and getting back to designing quickly. In this guide, we will explore the primary reasons behind this issue, step-by-step solutions, and best practices to prevent it from happening again.

## Why Are Solid Tools Greyed Out in Fusion 360?

Several factors can cause solid tools in Fusion 360 to become inaccessible. Usually, the issue is tied to the current workspace, selection mode, or the state of your design. Addressing these root causes can restore full functionality to your solid modeling tools.

### 1. Incorrect Workspace or Environment

Fusion 360 offers various workspaces such as Model, Patch, Render, Simulation, and CAM. Solid tools are only available when you are in the **Model** workspace, which is dedicated to creating and editing solid geometry.

- If you accidentally switch to another workspace like Patch or Render, the solid tools will grey out.
- Confirm you are in the **Design** tab, which is the default workspace for solid modeling.

### 2. Not in the Correct Design or Body Selection Mode

Fusion 360 relies on correct selection modes to operate on bodies and components.

- If you've selected a component or body in a way that deactivates the tools, solid features might be disabled.
- Make sure you are selecting the body or component directly and that no other features or bodies are active in a way that limits access.

### 3. Active Bodies or Components Locking Features

Sometimes, parts of your design are locked or in a state that prevents modifications.

- If a body is marked as "Read-Only" or locked in a component, solid editing tools will grey out.
- Check if the body or component is active and unlocked.

### 4. The Model Is in a Non-editable State (e.g., Derived or Imported Geometry)

Imported or derived geometry might be read-only, making certain tools unavailable.

- Solid tools generally work on native, editable bodies.
- Imported meshes or FFF files may not support direct solid editing until converted or converted to a BRep.

### 5. Body or Feature Selection Issues

Having multiple bodies selected might disable certain editing tools if the selection isn't suitable for the operation.

- Verify that you've selected only one body or component when needed.
- Clear unintended selections or switch to the correct selection.

### 6. The Design Has Been Set to a Non-Editable State

Fusion 360 offers different design states, like 'Design History' being turned off.

- When the Design History is disabled, some features may be limited or require special toggling to access.
- Enable design history for full functionality by clicking the "Edit Timeline" button if needed.

## 7. Software Glitches or Bugs

Occasionally, Fusion 360 might experience temporary glitches causing the tools to be greyed out.

- Ensure your software is up-to-date.
- Restart Fusion 360 to clear temporary issues.
- Check for updates in Autodesk Account or via the app store.

## Step-by-Step Solutions to Reactivate Greyed-Out Solid Tools

Once you identify the cause, follow these practical steps to restore access to your solid tools.

### 1. Confirm You Are in the Correct Workspace

- Navigate to the "Design" workspace by clicking the workspace dropdown at the top-left corner.
- Ensure you are working within the **Model** environment for solid related features.

### 2. Check Your Selection

- Ensure that your cursor is selecting the correct geometry—bodies or faces, not sketches or construction geometry.
- Hover over bodies and confirm they are highlighted.

### 3. Unlock or Activate Locked Bodies or Components

- Right-click the locked body or component.
- Choose "Unlock" or "Activate" from the context menu.
- Confirm that the body/component is unlocked; it should now be editable.

## 4. Convert Imported Geometry to Solid

- If working with meshes or imported geometry:
- Use the "Mesh to BRep" tool available in the solids menu.
- This converts mesh data into a solid body, enabling full solid tools.

## 5. Check and Enable Design History

- If necessary, click the "Timeline" icon at the bottom to toggle design history back on.
- This allows for more advanced editing and feature creation.

## 6. Reset or Restart Fusion 360

- Save your work.
- Exit and restart Fusion 360.
- If the issue persists, consider resetting preferences or reinstalling the software.

## 7. Update Fusion 360 to the Latest Version

- Go to the Autodesk Desktop App.
- Check for updates and install any available patches.
- Software updates often resolve bugs that cause tools to be greyed out.

## Practical Tips and Best Practices

- Always save your work frequently.
- Keep your software updated to lessen bugs.
- Use the "Browser" and "Bodies" panel to verify which objects are active and unlocked.
- When importing models, convert meshes into BRep for full feature access.

- Regularly switch workspace back to "Design" before editing solid features.

## Comparing Fusion 360's Workspaces

| Workspace | Purpose                             | Supports Solid Tools | Notes                           |
|-----------|-------------------------------------|----------------------|---------------------------------|
| Model     | Creating and editing solid geometry | Yes                  | Default workspace for modeling  |
| Patch     | Surface modeling                    | No                   | Focused on surfaces, not solids |

|            |                             |    |                                        |
|------------|-----------------------------|----|----------------------------------------|
| Render     | Rendering and visualization | No | Not for modeling                       |
| Simulation | Analyzing designs           | No | Limited to simulation-related features |
| CAM        | Manufacturing setup         | No | Focuses on toolpaths and machining     |

## Conclusion

Encountering greyed-out solid tools in Fusion 360 is a common frustration, but with a systematic approach, it is generally easy to resolve. Usually, the root cause lies in workspace selection, object locking, or model state. By confirming you are in the correct environment, verifying your selections, unlocking bodies, and converting imported geometry when needed, you can restore full access to Fusion 360’s powerful solid modeling tools. Keeping your software up-to-date and

following best practices will also prevent future issues, helping you maintain a smooth and productive design process.

## FAQ

### 1. Why are my solid tools disabled in Fusion 360?

**Ans:** They are disabled because you are in the wrong workspace, the selected bodies are locked, or your geometry is not in an editable state.

### 2. How can I unlock a body that's greyed out?

**Ans:** Right-click the body or component in the browser and select "Unlock" or "Activate."

### 3. Can imported meshes be edited with solid tools?

**Ans:** Not directly; you need to convert the mesh to a BRep or solid body using the "Mesh to BRep" tool.

### 4. Why do solid tools sometimes become greyed out when I restart Fusion 360?

**Ans:** This may be due to temporary glitches; restarting the software or updating it often resolves such issues.

### 5. How do I switch back to the Design workspace?

**Ans:** Use the workspace dropdown menu at the top-left corner and select "Design."

### 6. What is the role of design history, and how does it affect solid tools?

**Ans:** Design history records your feature creation timeline; disabling it may limit some editing capabilities, so enable it for full access.

### 7. How do I convert imported geometry into a solid body?

**Ans:** Use "Mesh to BRep" within the solids menu to convert imported meshes into editable solid bodies.

# 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.

# Practice What You've Learned

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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:

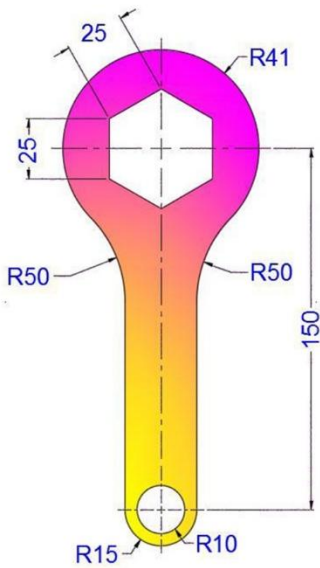
- ✓ Practice exercises from the book
- ✓ A brief overview of the complete book
- ✓ Options to explore or request the full sample

**Your hands-on Fusion 360 practice starts next.**

# AUTODESK FUSION 360 ALL IN ONE WORKBOOK

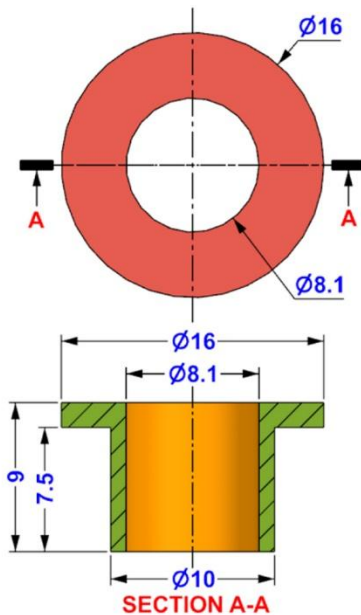
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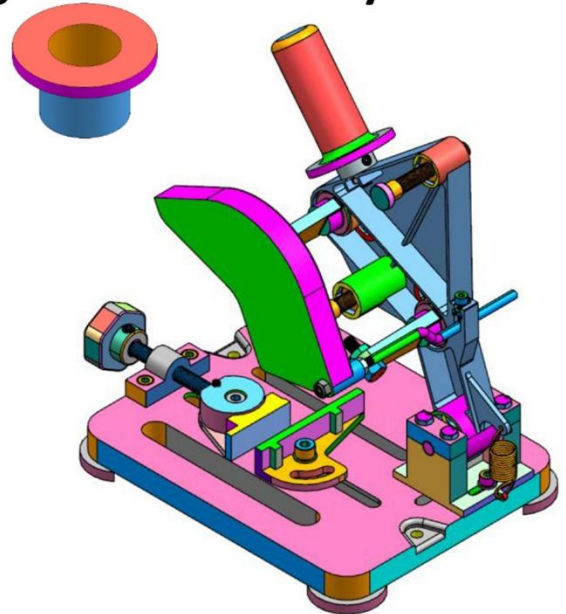
2D Sketching

- 3D Modeling



3D Modeling

- Assembly



Assembly

SACHIDANAND JHA

# AUTODESK FUSION 360 ALL IN ONE WORKBOOK

500+ PRACTICE EXERCISES

2D Sketching • 3D Modeling • Assembly Drawings

SACHIDANAND JHA



Dear Reader,

Thank you for choosing the AUTODESK FUSION 360 ALL IN ONE WORKBOOK. This book is part of the CADIN360° learning series, created to help engineers, students, and professionals master Fusion 360 through structured and practical exercises.

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:**

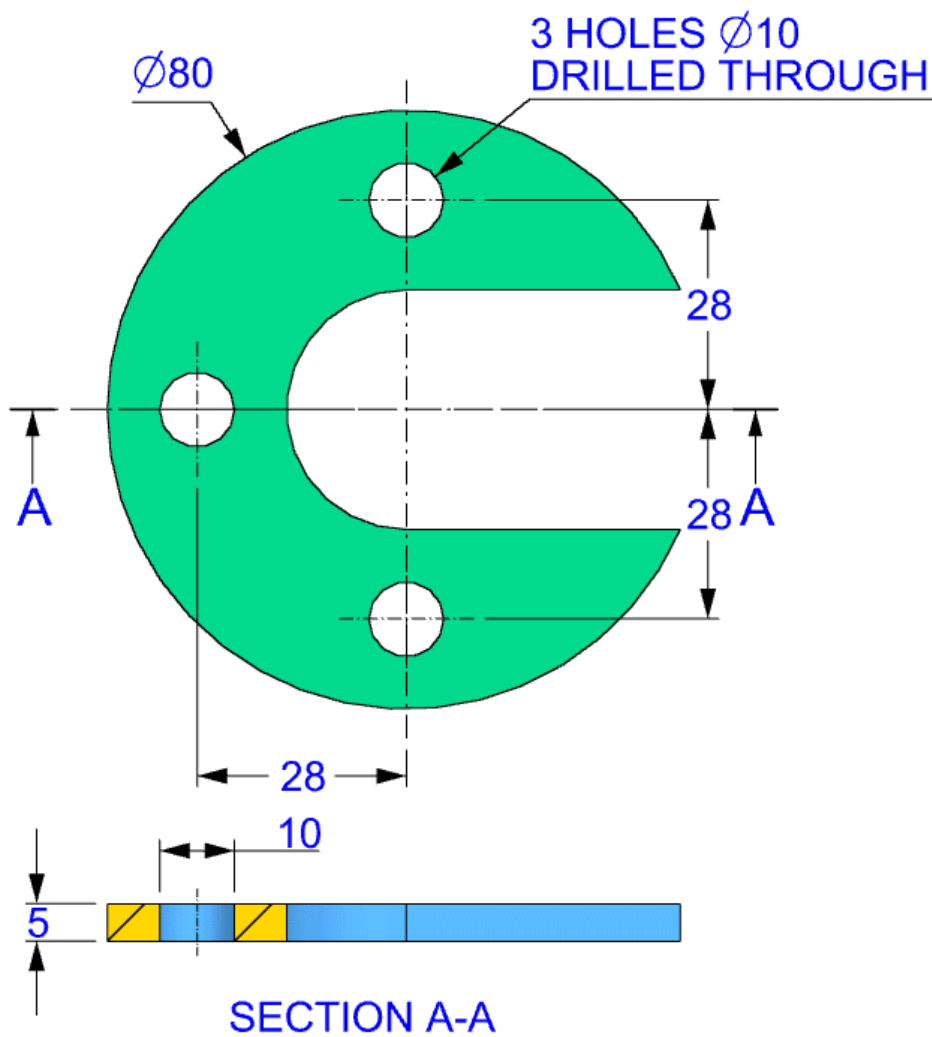
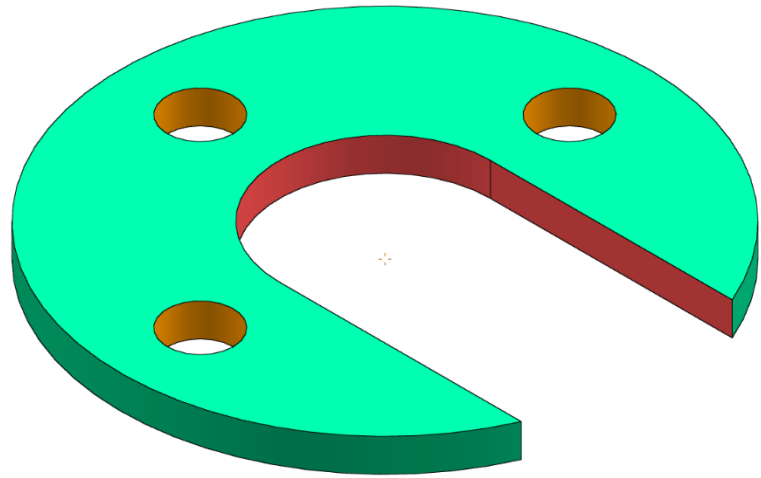
- Complete the exercises in order, or jump to any skill level you prefer.
- All dimensions are in millimeters.
- Where dimensions are missing, apply logic or practice estimation.
- This book is ideal for both students and professionals preparing for industry design work.

### **Note:**

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

Happy learning!  
– Team CADIN360

3D

**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

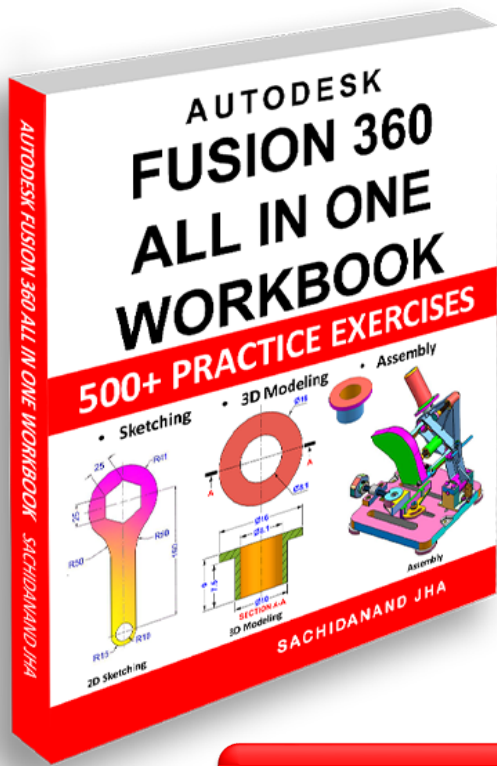
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- Same quality as our professional training material
- Compatible with the latest software version

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The **complete sample PDF will be delivered automatically**  
after the form is submitted.

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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
- ✓ Get 200 3D Exercises in .f3d file format
- ✓ Get All Assembly Exercises in .STP file
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# Thank You for Learning with Us!

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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)