

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



HOW TO CREATE FIRST JOINT IN FUSION 360

• LEARN • • APPLY • • GROW •

Introduction

Creating the first joint in Fusion 360 is a fundamental skill that every designer and engineer needs to master. Joints are critical for building functional assemblies, enabling parts to move realistically or stay fixed together. Whether you're designing a mechanical linkage, a mechanical arm, or just practicing the basics of Fusion 360, understanding how to create a joint is essential. In this guide, we will walk through the entire process—step by step—so you can confidently make your first joint in Fusion 360, optimize your workflow, and eventually tackle more complex assemblies.

Understanding Fusion 360 Joints: The Basics

Before diving into the actual steps, it's important to understand what joints are in Fusion 360. Joints are constraints that connect two components, allowing relative movement or fixing parts together. Fusion 360 supports various types of joints, including Rigid, Revolute, Slider, Cam, Pin Slot, and Ball joints. Knowing which type to use depends on your design requirements.

Why Use Joints in Fusion 360?

- To simulate real-world mechanical movements
- To assemble components quickly and accurately
- To test prototyping ideas in a virtual environment
- To facilitate assembly instructions or manufacturing processes

Having a clear understanding of your intended function guides your choice of joint.

Preparing Your Components for Joints

Good joint creation starts with proper component preparation. Follow these tips before creating your first joint:

- 1. Model components accurately – Ensure parts are complete with correct dimensions.**
- 2. Create components as separate bodies – This simplifies assembly and joint creation.**

3. **Use consistent naming conventions – Helps identify parts easily during joint selection.**
4. **Position components roughly in the desired working location – Precise positioning isn't necessary initially; joints will define exact placement.**

Now, let's start with the actual process of creating your first joint in Fusion 360.

Step-by-Step Guide to Creating Your First Joint in Fusion 360

1. Open or create your assembly workspace

- Launch Fusion 360.
- Open an existing project or create a new design.
- Ensure each part you want to join is modeled as a separate component.

2. Position components roughly

- Use the Move tool to position parts in a logical location close to where the joint will be placed.
- This step isn't precise; the joint will be used to define exact positioning.

3. Activate the Assemble menu

- In the toolbar, click on **Assemble**.
- From the dropdown, select **Joint** or **As-built Joint** based on your needs.

4. Select the first component

- Fusion 360 will prompt you to select the first component. Click on the component you want to act as the base or fixed part.

5. Select the second component

- Click on the second component to be connected.
- Fusion 360 will now display small yellow icons indicating possible joint origins.

6. Pick the joint origins

- Hover over the components to select the specific faces, edges, points, or features where the joint will be attached.
- Common choices include cylindrical faces for revolute joints or flat faces for slider joints.

7. Adjust joint placement

- After selecting the origins, Fusion 360 will preview the joint.
- Use the move or rotate handles to fine-tune the position if necessary.

8. Select and assign the joint type

- In the Joint dialog box, choose the appropriate joint type:

Joint Type	Description	Use Case Examples
Rigid	No movement	Fixed parts

Revolute	Rotational movement	Gears, hinges
Slider	Linear sliding movement	Pistons, drawer slides
Ball	Multi-axis rotation	Spherical joints

- Choose a type based on your design intent.

9. Define the motion or fix position

- Set joint limits if necessary.
- For fixed parts, choose **Rigid**.
- For movable parts, specify the degrees of freedom.

10. Confirm and finish

- Click **OK** to create the joint.
- Fusion 360 will now treat these components as connected, either fixed or with motion depending on the joint type.

Practical Example: Creating a Revolute Joint for a Hinged Door

Suppose you're designing a door hinge:

1. **Model the door and the hinge as separate components.**
2. **Roughly position the hinge near the edge of the door.**
3. **Use the Joint command.**
4. **Select the hinge's pin as the first component.**
5. **Select the door as the second component.**
6. **Choose the cylindrical face of the hinge pin and the edge of the door.**
7. **Select Revolute as the joint type.**
8. **Adjust the joint origin if needed and set limits to simulate hinge movement.**
9. **Complete the process by confirming the joint.**

This simple example demonstrates how joints enhance your design and simulate real-world mechanics.

Common Mistakes and How to Avoid Them

- **Incorrect component selection:** Always verify you've selected the right faces or features for the joint origins.
- **Misaligned parts:** Rough positioning saves time; precise assembly will be handled by joints.
- **Choosing wrong joint types:** Match the joint to your intended motion or fixity.
- **Ignoring joint limits:** Use limits to prevent unrealistic movements.

Training yourself to double-check each step ensures a smooth workflow.

Pro Tips for Creating Effective Joints in Fusion 360

- Use **As-Built Joints** to connect components that are already in correct position.
- When creating multiple joints, do so systematically to avoid confusion.
- Create visual guides or sketches to mark joint locations before assembling.
- Use **Rigid** joints for fixed parts, and only use movable joints when necessary.
- Test joint movement early to ensure it behaves as expected before progressing further.

Comparing Fusion 360 Joints: Which One to Use?

Joint Type	Purpose	Typical Use Case	Flexibility
Rigid	Fixed connection	Assembled parts that don't move	None
Revolute	Rotational movement	Hinges, rotating arms	Rotates around a single axis

Slider	Linear movement	Pistons, sliding drawers	Moves along a straight line
Ball	Multi-axial rotation	Spherical joints, universal joints	Rotates in multiple directions

Choosing the right joint type helps in accurately modeling real-world mechanisms.

Conclusion

Creating your first joint in Fusion 360 is a foundational step in building complex assemblies and simulating functional designs. By understanding the basics, following a systematic approach, and practicing with real-world examples, you can master joint creation in Fusion 360 with confidence. Remember to select the appropriate joint type, accurately choose the origins, and fine-tune the placement for optimal results. As you gain experience, you'll unlock more advanced assembly techniques that expand your design capabilities.

FAQ

1. How do I create a fixed joint in Fusion 360?

Ans : Select the components, then choose the Rigid joint type to fix parts together without movement.

2. Can I change a joint type after creating it?

Ans : Yes, you can edit the joint in the Browser by right-clicking the joint and selecting **Edit Joint** to change its type or properties.

3. What is the difference between Assembly and As-Built Joint in Fusion 360?

Ans : **Assembly joints** are created between components that are moveable, while **As-Built Joints** are used to connect components that are already positioned without the need for adjustments.

4. How do I test the movement of a joint in Fusion 360?

Ans : Use the **JS** (Joint Study) feature to animate and analyze joint movement within your assembly.

5. Why is my joint not moving as expected?

Ans : Possible reasons include incorrect joint type selection, improper origin placement, or conflicting joints. Review the joint setup for accuracy.

6. Can I create multiple joints between the same components?

Ans : Yes, you can create multiple joints, but it's best to plan their positions carefully to prevent conflicts.

7. Is it possible to animate joints in Fusion 360?

Ans : Yes, Fusion 360 allows you to animate joints to simulate movement during visualization or simulation purposes.

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:

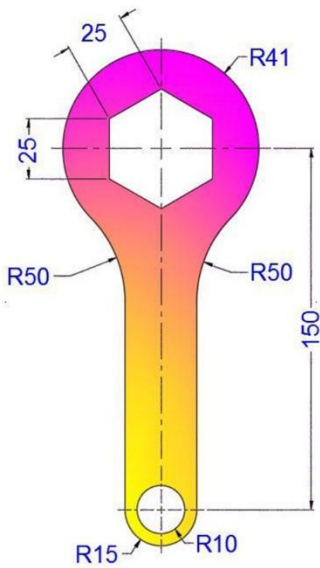
- ✓ Practice exercises from the book
- ✓ A brief overview of the complete book
- ✓ Options to explore or request the full sample

Your hands-on Fusion 360 practice starts next.

AUTODESK FUSION 360 ALL IN ONE WORKBOOK

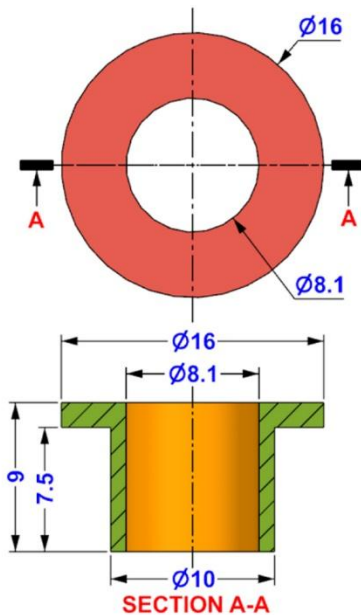
500+ PRACTICE EXERCISES

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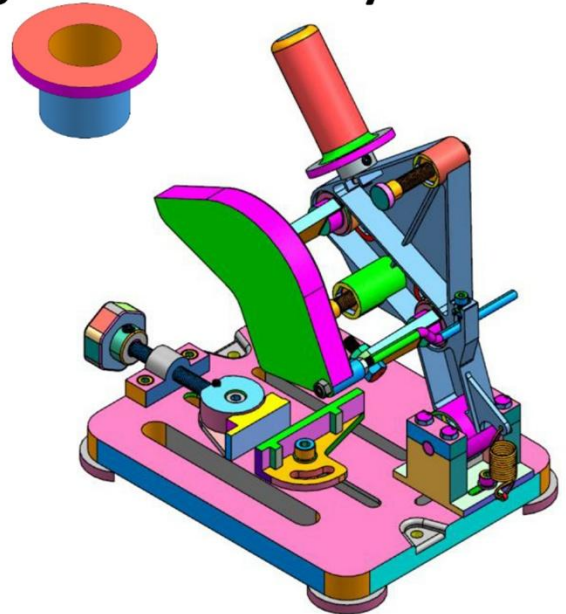
2D Sketching

- 3D Modeling



3D Modeling

- Assembly



Assembly

SACHIDANAND JHA

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2D Sketching • 3D Modeling • Assembly Drawings

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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



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:

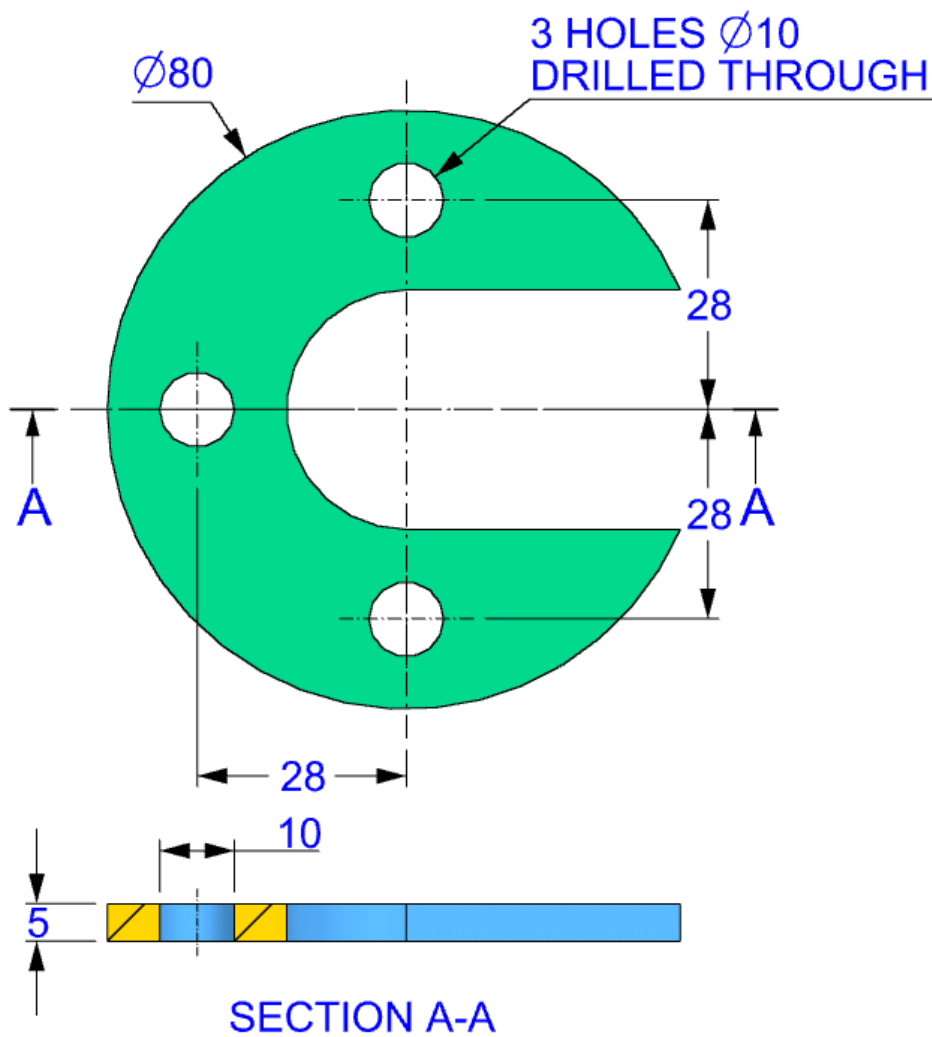
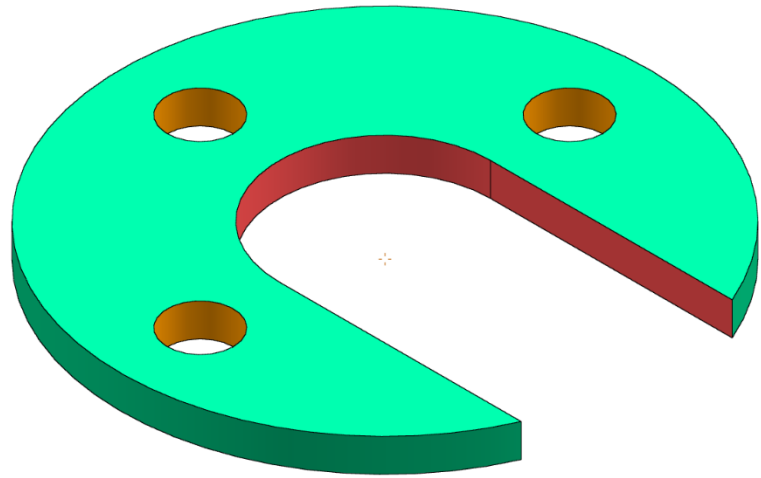
- Complete the exercises in order, or jump to any skill level you prefer.
- All dimensions are in millimeters.
- Where dimensions are missing, apply logic or practice estimation.
- This book is ideal for both students and professionals preparing for industry design work.

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

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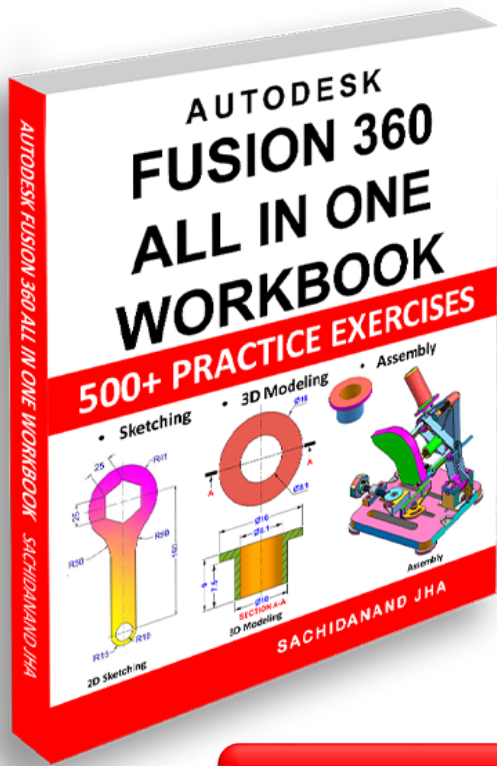
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What's Included in the **FUSION 360 ALL IN ONE WORKBOOK?**

- ✓ 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
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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 3rd Angle projection
- All dimensions are in mm(metric system)