

SOLIDWORKS

2026

BLOG

A Note to Our Readers

2026

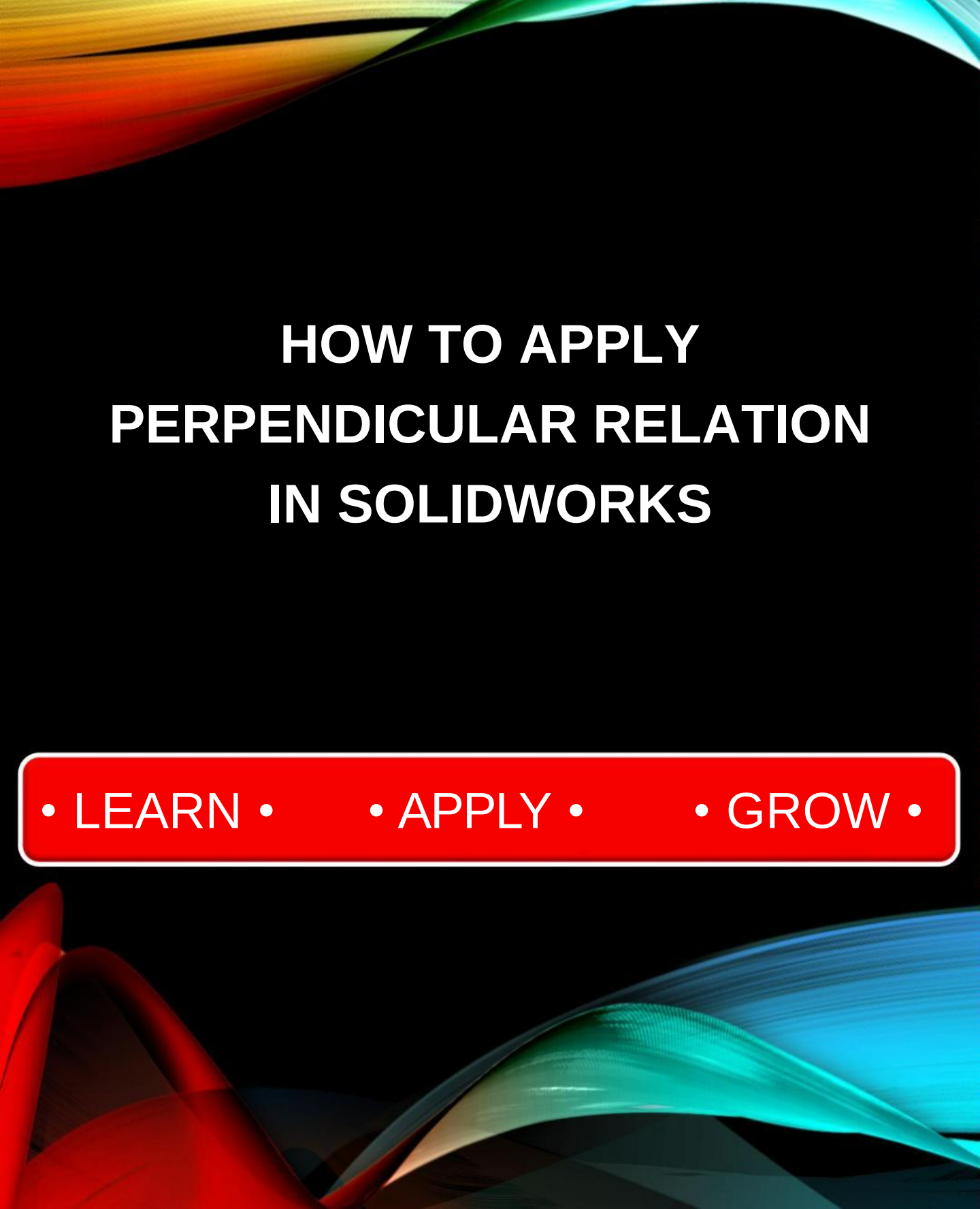
This blog has been created using a combination of artificial intelligence tools and human review to help deliver clear, structured, and up-to-date learning content.

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 APPLY PERPENDICULAR RELATION IN SOLIDWORKS

• LEARN • • APPLY • • GROW •

Introduction

Applying a perpendicular relation between components or features in SolidWorks is a fundamental skill for creating precise and accurate designs. Whether you're designing mechanical parts, assemblies, or complex assemblies, establishing perpendicular constraints ensures proper alignment and optimal function. In this comprehensive guide, we will walk you through the step-by-step process of applying perpendicular relations in SolidWorks. You'll learn how to do it effectively, common pitfalls to avoid, and best practices to streamline your workflow. Mastering perpendicular constraints not only improves your design accuracy but also enhances your proficiency in SolidWorks—making your engineering tasks more efficient and professional.

Understanding Perpendicular Relations in SolidWorks

Before diving into how to apply perpendicular relations, it's important to clarify what they are and why they matter in 3D CAD design.

A perpendicular relation in SolidWorks means fixing the angle between two selected entities—like lines, edges, or planes—at 90 degrees. This constraint ensures that the features or components are exactly orthogonal, which is crucial in mechanical design, ensuring correct assembly, movement, and functionality.

Common scenarios for using perpendicular relations include:

- Aligning holes in different faces
- Ensuring hinges operate at right angles
- Assembling gears, shafts, or brackets with precise orthogonal positioning
- Creating accurate sketches with right-angle constraints

Having a solid grasp of how to apply these relations keeps your models robust and reduces errors during manufacturing.

How to Apply Perpendicular Relation in SolidWorks

Applying a perpendicular relation in SolidWorks can be achieved primarily during sketching or assembly constraints. Here's a detailed step-by-step guide for both contexts.

Applying Perpendicular Relation in Sketch Mode

Using perpendicular constraints within sketches is fundamental for constructing accurate 2D profiles.

1. Begin a new Sketch

- Select the face or plane where you want to sketch.
- Click on the **Sketch** tool from the CommandManager and choose the appropriate plane.

2. Create the entities to be constrained

- Draw two lines or points that you want to set at right angles.
- Ensure both entities are visible and selectable.

3. Select the entities

- Click on the first line or entity.
- Hold the **Ctrl** key and click on the second line or entity.

4. Apply the perpendicular relation

- With both entities selected, open the **Add Relations** menu.
- Click on **Perpendicular** from the list of relation options.
- The sketch entities will now be constrained at a 90-degree angle.

5. Confirm and test

- Exit the relation feature.
- Drag the entities slightly to verify that the perpendicular relation holds firm.

- Complete your sketch for further operations.

Applying Perpendicular in Assembly Mode

Perpendicular constraints in assemblies are crucial for positioning parts correctly relative to each other.

1. Insert the components

- Open or create your assembly file.
- Insert the parts you want to align perpendicularly.

2. Use Mates for perpendicular relation

- Click on **Mate** from the assembly toolbar.
- Select the face, edge, or axis of the first component.
- Hold **Ctrl** and select the face, edge, or axis of the second component.

3. Choose the Perpendicular Mate

- In the Mate PropertyManager, select **Perpendicular**.
- SolidWorks automatically sets the two entities at a 90-degree relation.

4. Adjust and verify

- Use the **Preview** button to confirm the fit.
- Click **OK** to apply the mate.
- Test the movement to ensure the components stay perpendicular as designed.

Practical Examples of Applying Perpendicular Relations

Example 1: Creating a Bracket with Right-Angle Holes

Suppose you need to design a metal bracket with holes drilled at right angles to ensure proper mounting.

Steps:

- Sketch the bracket profile.
- Draw two lines representing the holes' axes.
- Apply perpendicular relations between these lines in the sketch.
- Use the hole wizard to position the holes aligned with these axes.

Example 2: Assembling a Shaft and Gear

To assemble a gear onto a shaft at a right angle:

- Insert the shaft and gear as separate components.
- Mate the shaft's axis to the gear's hole axis.
- Apply a **perpendicular** mate between the gear face and the shaft's end to ensure orthogonal positioning.

Example 3: Designing a Mechanical Arm with Orthogonal Joints

- Sketch the arm components.
- Use perpendicular relations to align joint axes.
- Assemble the parts by selecting axes or faces, then applying perpendicular mates.

Common Mistakes When Applying Perpendicular Relations

Avoid these frequent pitfalls:

- **Selecting incompatible entities:** Make sure you're selecting the correct entities (lines, edges, axes).
- **Applying perpendicular relations in 3D where not needed:** Sometimes, a 2D sketch relation suffices; over-constraining can cause issues.
- **Not verifying after applying:** Always test the constrained entities to ensure the relation holds under movement or editing.
- **Ignoring existing constraints:** Previous relations can conflict or over-constrain your sketch or assembly.

Tips and Best Practices for Using Perpendicular Relations

- Use **snap points** or **construction geometry** to facilitate precise alignment.
- When constraining in sketches, combine perpendicular relations with coincident and horizontal/vertical relations for more controlled geometry.
- In assemblies, pre-plan the sequence of mates to avoid over-constraint.
- Keep your sketches and assemblies simple; add relations gradually.
- Regularly test the movement or edits to check for unintended constraints.

Comparing Sketch and Assembly Perpendicular Constraints

Aspect	Sketch Perpendicular Relation	Assembly Perpendicular Mate
Purpose	Creates orthogonal geometry during sketching	Positions components at right angles in an assembly
Application	During 2D sketch creation	During 3D component positioning
Constraints	Fixed on geometry, part of sketch relations	Mates that define component relationships

Flexibility	Limited to sketch plane	Can be adjusted during assembly to modify position
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Conclusion

Applying perpendicular relations in SolidWorks is a powerful technique that ensures precision and proper alignment in your designs. Whether working within sketches or during the assembly process, mastering these constraints simplifies complex modeling tasks, reduces errors, and improves manufacturability. Practice applying perpendicular constraints in various scenarios to enhance your SolidWorks proficiency and create more accurate, professional models.

FAQ

1. How do I apply a perpendicular relation in a sketch in SolidWorks?

Ans : Select two sketch entities, open the Relations menu, and click on "Perpendicular."

2. Can I change or remove a perpendicular relation once it's applied?

Ans : Yes, select the relation in the sketch or feature manager, then delete or modify it as needed.

3. How do I ensure parts remain perpendicular during assembly?

Ans : Use the **Perpendicular Mate** between relevant faces, edges, or axes to fix their right-angle relation.

4. What are common mistakes when applying perpendicular constraints?

Ans : Selecting incompatible entities, over-constraining, or not verifying the relation's effectiveness afterward.

5. Is it possible to apply perpendicular relations to curved surfaces?

Ans : Perpendicular relations are typically used with straight edges or axes; curved surfaces require different constraints like tangent or coincident relations.

6. How can I troubleshoot if a perpendicular relation isn't holding?

Ans : Check for conflicting constraints, ensure the correct entities are selected, and verify that the relation is active and unbroken.

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 SolidWorks.

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

2026

You've just completed this blog and learned important concepts in SolidWorks.

To help you practice and apply what you've learned, the next pages include a sample from our SolidWorks book .This sample contains practice exercises and real-world practice tasks designed to strengthen your skills.

What you'll find next:

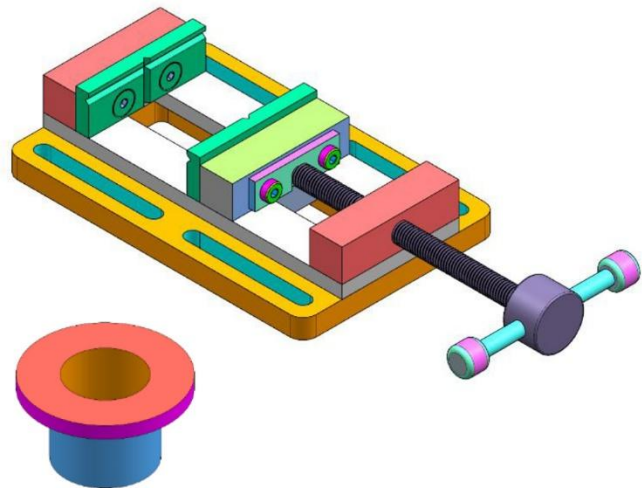
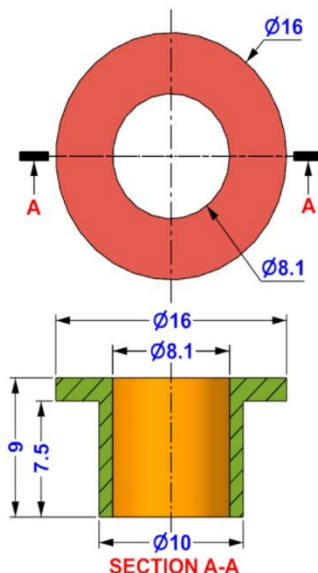
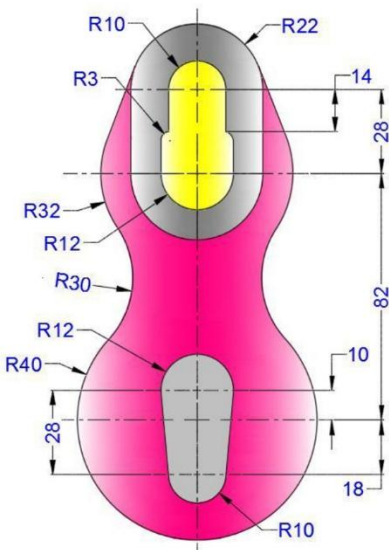
- ✓ Practice exercises from the book
- ✓ A brief overview of the complete book
- ✓ Options to explore or request the full sample

Your hands-on SolidWorks practice starts next.

SOLIDWORKS ALL IN ONE WORKBOOK

500+ PRACTICE EXERCISES

- Sketching
- 3D Modeling
- Assembly



SACHIDANAND JHA

SOLIDWORKS ALL IN ONE WORKBOOK

500+ PRACTICE EXERCISES

• 2D Sketching • 3D Modeling • Assembly Drawings

SACHIDANAND JHA



Dear Reader,

Thank you for choosing the **SOLIDWORKS ALL IN ONE WORKBOOK**. This book is part of the CADIN360° learning series, created to help engineers, students, and professionals master SolidWorks 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

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Customer feedback is critical to our efforts at CADIN360.

Best regards,

Sachidanand Jha
Founder & CEO, CADIN360



SOLIDWORKS ALL IN ONE WORKBOOK

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SOLIDWORKS 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 **SOLDIWORKS** and also suitable for Other Feature-Based Modeling Software such as Inventor, Catia, Fusion 360, NX, Solid Edge, AutoCAD, PTC Creo etc.
- ❖ Designed for students, engineers, drafters, and designers looking for extensive CAD practice using SolidWorks.
- ❖ 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 SOLIDWORKS 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:

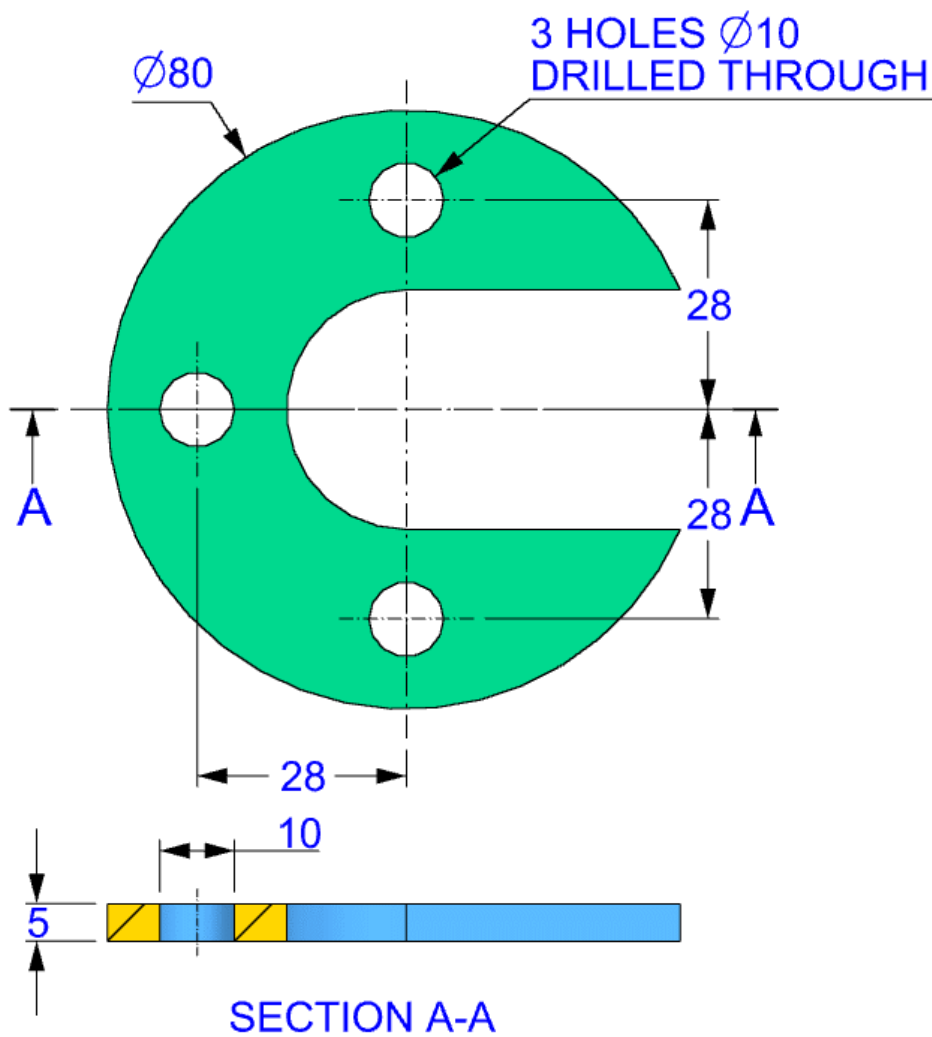
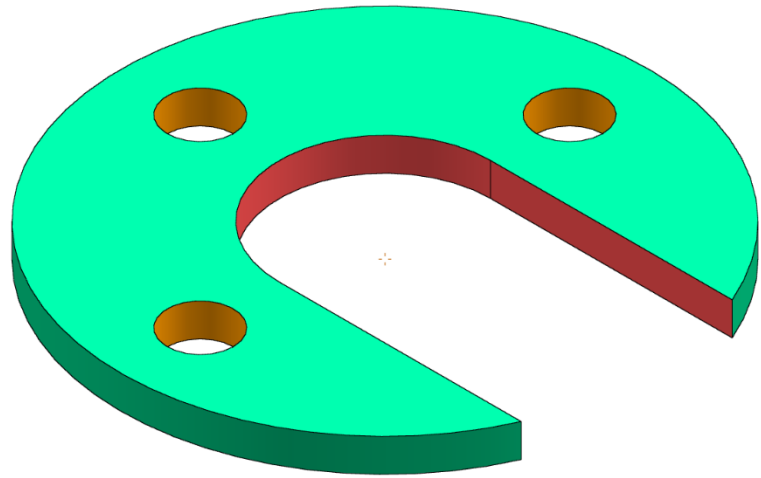
- Complete the exercises in order, or jump to any skill level you prefer.
- All dimensions are in millimeters.
- Where dimensions are missing, apply logic or practice estimation.
- This book is ideal for both students and professionals preparing for industry design work.

Note:

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

Happy learning!
– Team CADIN360

3D

EXERCISE-01

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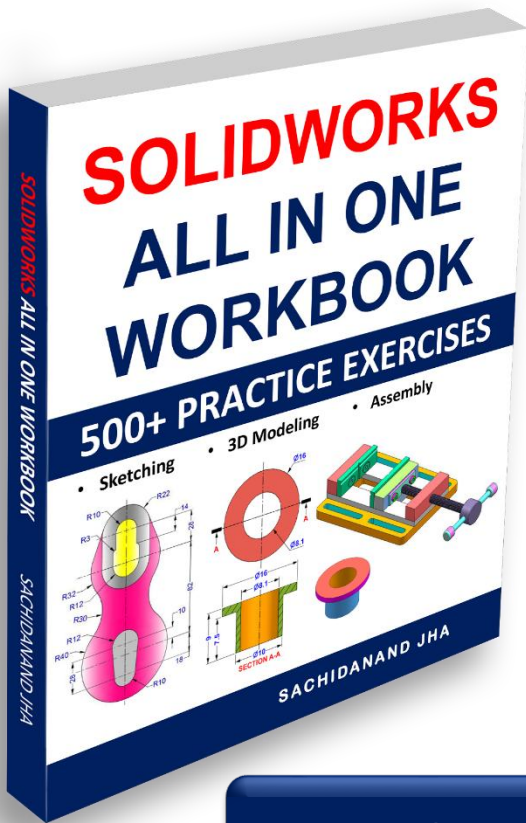
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What's Included in the **SOLIDWORKS ALL IN ONE WORKBOOK?**

- ✓ Books contains exercises of Sketching, 3D Modeling & Assembly.
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We're honored to be a part of your journey.

Happy Designing!

– Team **Cadin360**

Master **SolidWorks** with Real-World Practice Exercises

This book contains over 500 SolidWorks practice exercises including sketching, 3D modeling, and assembly drawings.

Designed for students, engineers, and professionals to build practical CAD modeling skills.

SOLIDWORKS 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)