

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 VERTICAL RELATION IN SOLIDWORKS

• LEARN • • APPLY • • GROW •

Introduction

Applying vertical relations in SolidWorks is a fundamental skill that enhances the precision and functionality of your 3D models. Whether you're designing mechanical assemblies or creating complex parts, mastering vertical constraints helps ensure your components align correctly along a specific axis. This tutorial will walk you through the step-by-step process of applying vertical relations in SolidWorks, providing practical examples, common mistakes to avoid, and best practices for efficient modeling. By the end, you'll have the knowledge to confidently use vertical relations to improve your design accuracy and streamline your workflow.

Understanding Vertical Relationship in SolidWorks

Before diving into the steps, it's essential to understand what the vertical relation signifies within SolidWorks.

A "vertical relation" constrains two or more entities—such as points, lines, or surfaces—to align along a common vertical axis, typically the Z-axis in most coordinate systems. This relation is crucial when you want parts to stay aligned vertically without any skew or lateral displacement.

In SolidWorks, applying a vertical relation ensures that selected features maintain a strict vertical alignment, which is vital in assemblies, structural frames, or when creating parametric models with precise alignments.

How to Apply Vertical Relation in SolidWorks: Step-by-Step Guide

Applying vertical relations is straightforward but requires careful selection of entities and understanding of the command interface.

1. Prepare Your Sketch

- Open your part or assembly file in SolidWorks.
- Initiate a new sketch on the plane or face where you want to establish the vertical relation.
- Draw the entities (points, lines, or other geometries) you wish to constrain.

2. Select Entities for the Vertical Relation

- Click to select the first entity (e.g., a point or line endpoint).
- Hold down the 'Ctrl' key and select the second entity.
- Ensure that these entities are capable of being constrained together and that they are properly defined.

3. Access the Add Relations Tool

- With the entities selected, look for the "Add Relations" button on the left sidebar or in the PropertyManager.
- Alternatively, right-click on one of the selected entities, navigate to "Relations," and choose "Vertical."

4. Apply the Vertical Relation

- Click "Vertical" from the list of available relations.
- The selected entities will now be constrained to align vertically.

5. Confirm and Complete the Sketch

- Check that a vertical relation symbol (a vertical constraint line) appears next to your constrained entities.
- To verify, try adjusting one entity—both should move in unison along the vertical axis.
- Finish your sketch by clicking "Exit Sketch" once the vertical relation is applied.

Practical Example: Aligning Two Points Vertically

Imagine you need to line up two points vertically for a structural frame:

- Draw two points on your sketch.

- Select the first and second point.
- Apply the "Vertical" relation.
- Adjust one point; both should move vertically together, maintaining the same X and Y coordinates.

Common Mistakes When Applying Vertical Relations

- **Incorrect entity selection:** Applying the relation between unrelated features can cause unexpected behavior.
- **Over-constraining:** Adding multiple conflicting constraints can lead to errors or over-defining your sketch.
- **Ignoring the projection plane:** Applying vertical relation in an incorrect sketch plane might not produce the expected alignment.

Tips and Best Practices for Using Vertical Relations

- **Use construction lines:** Draw vertical construction lines to help visualize and align entities before applying relations.
- **Combine with other constraints:** Use relations like coincident or parallel to complement vertical constraints for more control.
- **Validate constraints:** Always verify the constraints after applying by moving entities to ensure they behave as intended.
- **Parametrize your sketches:** Use dimensions alongside vertical relations for more flexible and adaptive models.

Comparing Vertical Relations with Other Constraints

Constraint Type	Purpose	Application Scenario	Key Characteristic
Vertical	Align entities vertically	Ensuring points or edges stay aligned along z-axis	Constrains movement along one axis
Horizontal	Align entities horizontally	To keep elements on the same transverse plane	Constrains movement perpendicular to vertical

Parallel	Keep lines parallel	For geometric consistency across features	Is directional but not position-specific
Coincident	Make points or lines share points	For sharing endpoints or centers	Fixes entities together

Understanding these distinctions helps you choose the right relation for your design goals.

Practical Applications of Vertical Relations in Real-World Models

- **Structural frameworks:** Ensuring columns or beams are perfectly aligned along a vertical axis.
- **Automotive or aerospace parts:** Aligning holes, brackets, or mounting points vertically for assembly consistency.
- **Product design:** Positioning components in a multi-layered assembly to maintain uniformity.

Conclusion

Applying vertical relations in SolidWorks is a vital skill for creating precise, well-aligned models. By carefully selecting entities, applying the "Vertical" relation, and verifying constraints, you can significantly improve your design accuracy and efficiency. Remember to combine vertical constraints with other relations and dimensions to achieve complex, robust models. Practice consistently to develop an intuitive understanding of when and how to best use vertical relations in your workflow.

FAQ

1. What is the primary purpose of applying vertical relations in SolidWorks?

Ans : The primary purpose is to align two or more entities along the same vertical axis, ensuring they stay vertically parallel in the model.

2. Can I apply vertical relations in 3D models or only in sketches?

Ans : Vertical relations are primarily applied within sketches to control 2D geometry, but they help position 3D features based on sketch constraints.

3. How can I verify that a vertical relation has been correctly applied?

Ans : After applying, you can move one entity to see if the other moves correspondingly along the vertical axis and look for the vertical relation symbol.

4. Is it possible to delete a vertical relation if I want to change my design?

Ans : Yes, select the constrained entities, open the "Display/Delete Relations" menu, and delete the vertical relation.

5. Can I apply multiple vertical relations between the same entities?

Ans : No, applying duplicate vertical relations between the same entities is redundant; only one is necessary for the alignment.

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:

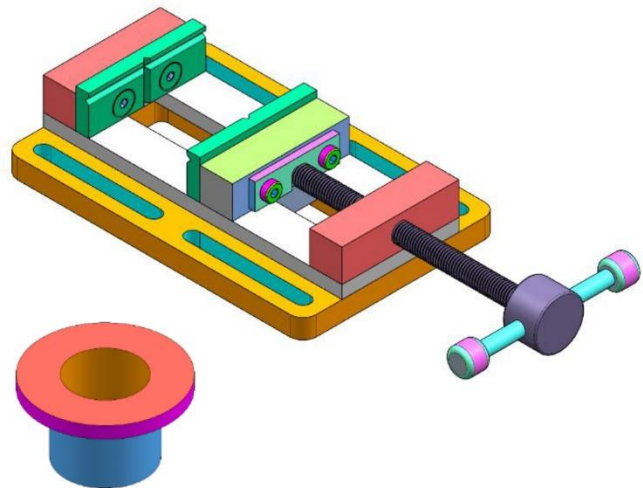
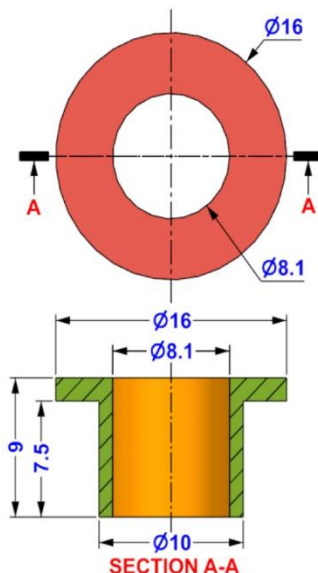
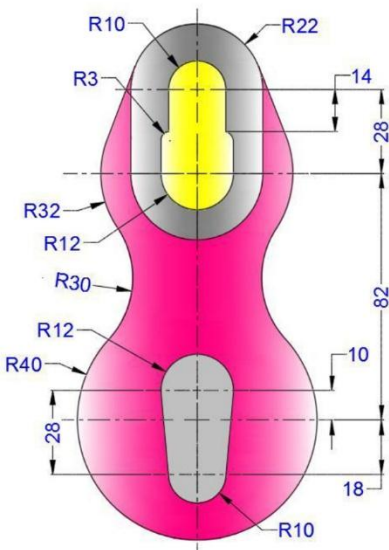
- ✓ Practice exercises from the book
- ✓ A brief overview of the complete book
- ✓ Options to explore or request the full sample

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

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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

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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

Published by CADIN360

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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:

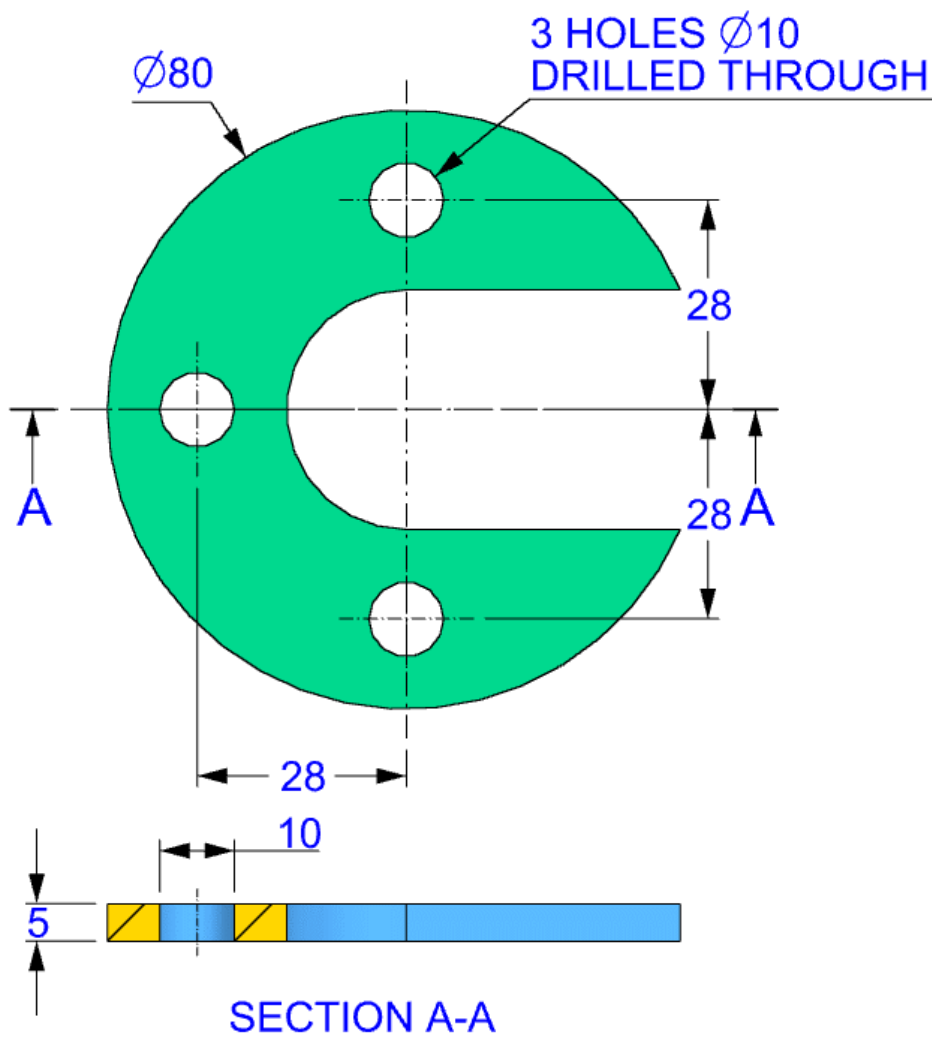
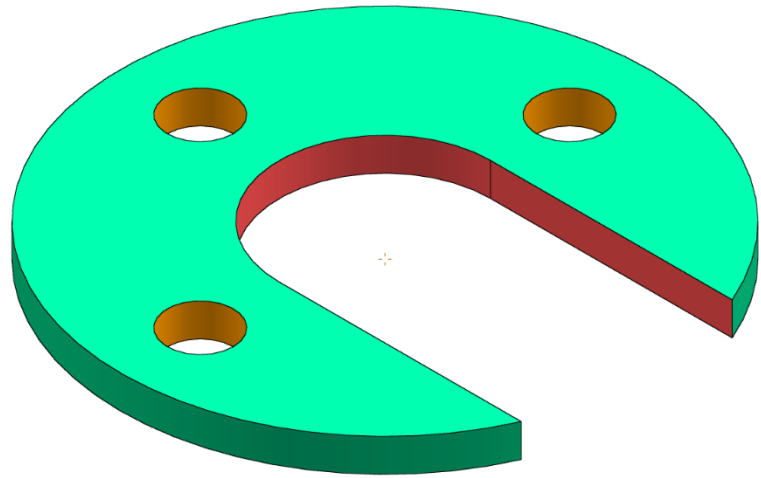
- Complete the exercises in order, or jump to any skill level you prefer.
- All dimensions are in millimeters.
- Where dimensions are missing, apply logic or practice estimation.
- This book is ideal for both students and professionals preparing for industry design work.

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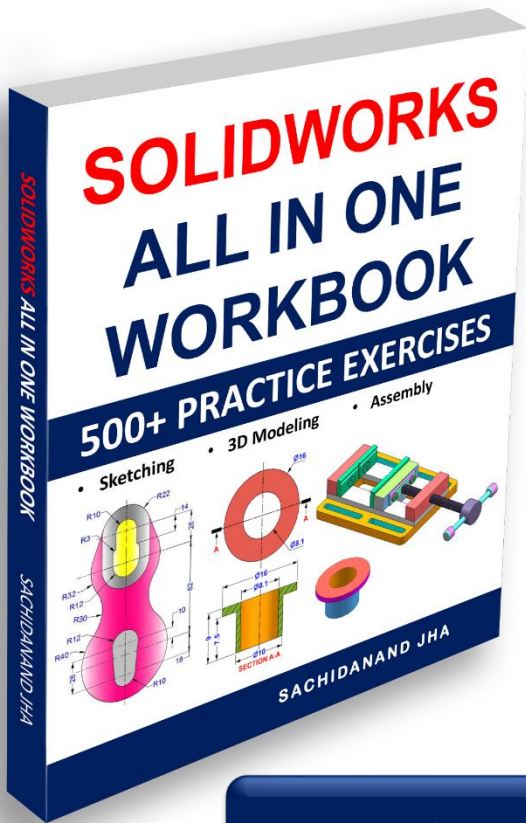
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What's Included in the **SOLIDWORKS ALL IN ONE WORKBOOK?**

- ✓ Books contains exercises of Sketching, 3D Modeling & Assembly.
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Happy Designing!

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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)