SOLIDWORKS

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.

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— CADIN360 Team

HOW TO UNDERSTAND SKETCH RELATIONS SIMPLY IN SOLIDWORKS

• LEARN • • APPLY • • GROW •

Introduction

Understanding sketch relations in SolidWorks is fundamental for creating precise, fully defined sketches. They help maintain geometric relationships between sketch entities, ensuring your parts behave predictably when modifications are made. Whether you're a beginner or looking to refine your skills, mastering sketch relations simplifies your design process and improves your productivity. This guide will break down how to understand sketch relations simply in SolidWorks, providing practical steps, real-world examples, common mistakes to avoid, and expert tips for efficient modeling.

What Are Sketch Relations in SolidWorks?

Sketch relations, also known as geometric constraints, are rules that define how sketch entities (lines, circles, points, etc.) relate to each other. They ensure that certain conditions are maintained as you modify the sketch, such as keeping two lines parallel or a point on a circle. These relations are essential for creating robust sketches that adapt well to changes, reducing errors and enhancing design intent.

Why are Sketch Relations Important?

- They help in creating fully defined sketches quickly.
- They improve the stability and predictability of your models.
- They facilitate parametric design, enabling easy modifications.
- They prevent accidental overlaps or misalignments during modeling.

How to Understand Sketch Relations Simply in SolidWorks

Grasping sketch relations might seem overwhelming initially. However, following a logical, step-by-step approach makes it straightforward. Here’s how to understand and effectively utilize sketch relations in SolidWorks:

Step-by-step Guide to Managing Sketch Relations

1. Creating a Basic Sketch

- Open SolidWorks and start a new part.
- Select a plane (Front, Top, or Right) and start a new sketch.
- Draw basic entities such as lines, circles, or points relevant to your design.

2. Selecting Entities for Relation Application

- Click on one or multiple sketch entities to apply relations.
- Use the **Select** tool and hold `Ctrl` to select multiple elements.

3. Applying a Sketch Relation

- With entities selected, go to the **Sketch** tab.
- Click **Add Relations** or right-click and choose **Add Relation**.
- Pick the relation type from the list that appears; common relations include:
- Coincident
- Parallel
- Perpendicular
- Tangent
- Horizontal or Vertical
- Equal
- Confirm by clicking **OK**.

4. Understanding Popular Sketch Relations

- **Coincident:** Makes a point lie on another entity (point on line, point on circle).

- **Parallel & Perpendicular:** Defines the angles between lines.
- **Tangency:** Ensures a circle or arc remains tangent to a line or another arc.
- **Horizontal & Vertical:** Fixes lines along the axes.
- **Equal:** Makes selected lines or circles have the same size or radius.

5. Analyzing Existing Relations

- Use the **Display/Delete Relations** tool to see all relations associated with selected entities.
- Relations are displayed as symbols next to entities.

6. Editing or Removing Relations

- To modify a relation, select it from the list and click **Delete** to remove or change it.
- Remember, excessive relations can over-constrain a sketch, causing errors.

Practical Example: Making a Fully Constrained Rectangle

1. **Draw a rectangle using four lines.**
2. **Add horizontal and vertical relations to align sides.**
3. **Use Equal relation on the length of opposite sides.**
4. **Add dimensions (e.g., length, width) to fix the size.**
5. **Verify that the sketch is fully defined (all black).**

Common Mistakes to Avoid

- Over-constraining sketches, leading to conflicts and errors.
- Relying solely on dimensions instead of relations for simplicity.
- Forgetting to delete unnecessary or redundant relations.
- Not verifying sketch "fully defined" status, which may result in unintended geometry.

- Mixing conflicting relations (e.g., parallel and perpendicular on the same line).

Tips and Best Practices for Managing Sketch Relations

- Always aim for the minimal number of relations needed to define the sketch.
- Use dimensions alongside relations for clarity and flexibility.
- Regularly check the status bar for "Fully Defined" status.
- Use the **Display/Delete Relations** tool to review relations.
- Keep relations organized to avoid confusion.
- Practice by creating simple sketches with various relations before tackling complex models.

Real-World Example: Parametric Gear Design

Suppose you're designing a gear with specific tooth profiles. You could:

- Use circle entities for the gear body.
- Apply the **Equal** relation to ensure all teeth are the same size.
- Use **Tangent** to align teeth profiles with the gear circumference.
- Fix key points with **Coincident** relations to the gear center.

This example demonstrates how sketch relations help in maintaining design constraints effortlessly as parameters change.

Comparing Sketch Relations with Dimensions

Feature	Sketch Relations	Dimensions
Purpose	Define geometric relationships explicitly	Control size and position numerically
Flexibility	Allows parametric updates within relations	Easily change sizes; less flexible for geometry constraints
Use Case	Complex geometric constraints for stability	Precise size and location control

Compatibility	Often used together for robust sketches	Can function independently or with relations

Using both in tandem leads to flexible yet stable sketches, ideal for complex modeling needs.

Conclusion

Understanding sketch relations simply in SolidWorks empowers you to create more reliable, flexible, and easily modifiable models. By mastering how to apply, analyze, and manage these relations, you can streamline your design process, minimize errors, and enhance your parametric modeling skills. Start practicing with basic sketches, gradually incorporate relations, and always aim for a minimal, well-organized set of constraints to maintain clarity and control over your designs.

FAQ

1. What are the most common sketch relations in SolidWorks?

Ans : The most common sketch relations include Coincident, Parallel, Perpendicular, Tangent, Horizontal, Vertical, and Equal.

2. How can I check which relations are applied to a sketch entity?

Ans : Use the **Display/Delete Relations** tool to view all relations associated with selected sketch entities.

3. Why is my sketch not fully defined even with relations applied?

Ans : Some relations might conflict or be redundant; check the relation list and remove or adjust conflicting relations.

4. Can I edit or delete relations after applying them?

Ans : Yes, select the relation or the related entities, then use the **Display/Delete Relations** tool to modify or delete them.

5. How do I avoid over-constraining my sketch?

Ans : Apply only necessary relations and dimensions, and regularly check for the "Fully Defined" status to prevent conflicts.

6. Is it better to rely more on dimensions or relations?

Ans : It's best to use a combination; relations control geometry relationships, while dimensions define exact sizes, providing a flexible and stable sketch.

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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.

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Practice What You've Learned

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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.

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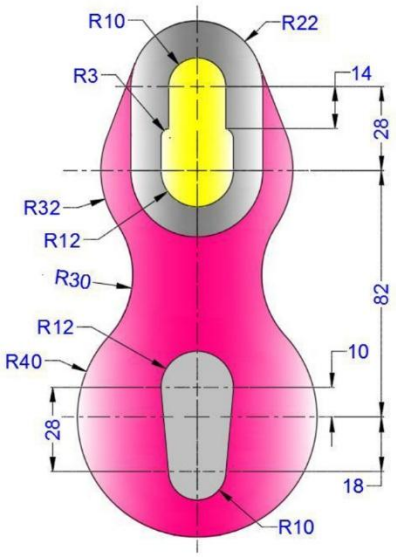
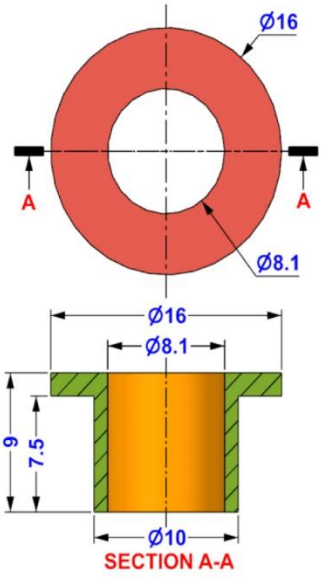
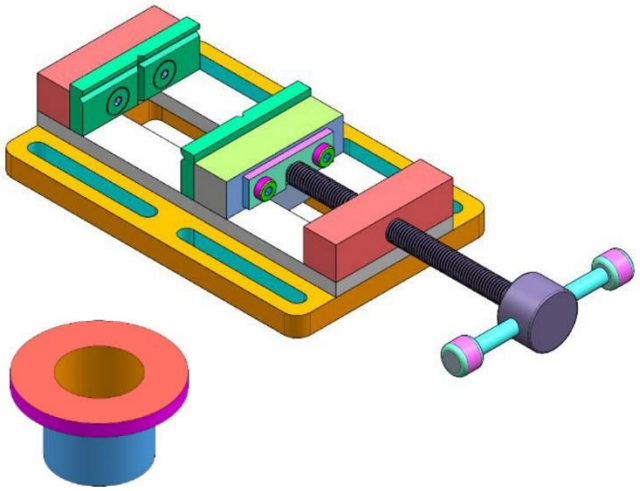
- Assembly







SACHIDANAND JHA

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500+ PRACTICE EXERCISES

• 2D Sketching • 3D Modeling • Assembly Drawings

SACHIDANAND JHA

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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.

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Best regards,

Sachidanand Jha
Founder & CEO, 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:

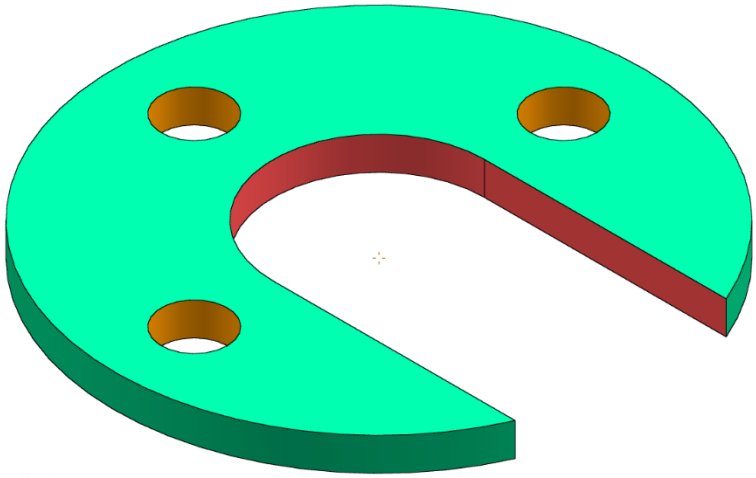
- Complete the exercises in order, or jump to any skill level you prefer.
- All dimensions are in millimeters.
- Where dimensions are missing, apply logic or practice estimation.
- This book is ideal for both students and professionals preparing for industry design work.

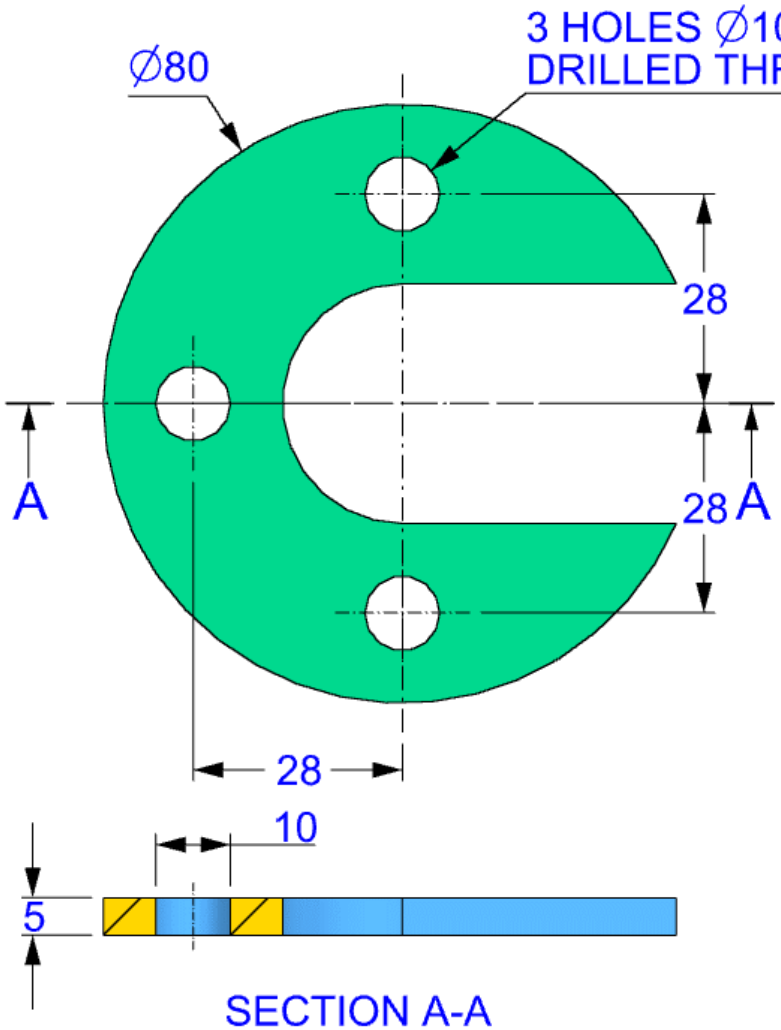
Note:

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

Happy learning!
– Team CADIN360

EXERCISE-01





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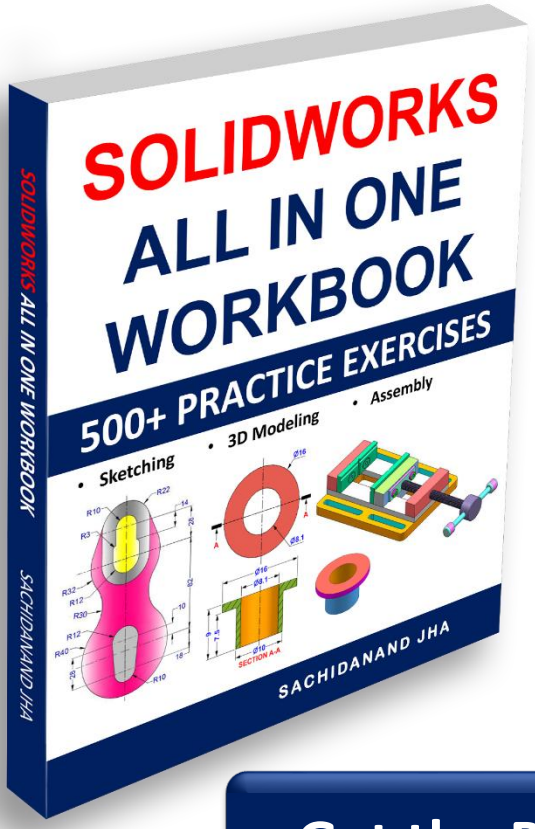

What's Included in the SOLIDWORKS ALL IN ONE WORKBOOK?

- ✓ 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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– Team **Cadin360**

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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)