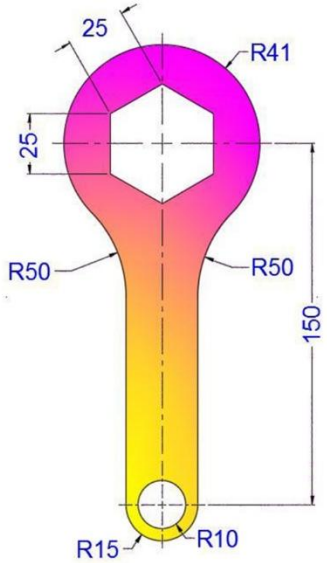
AUTODESK

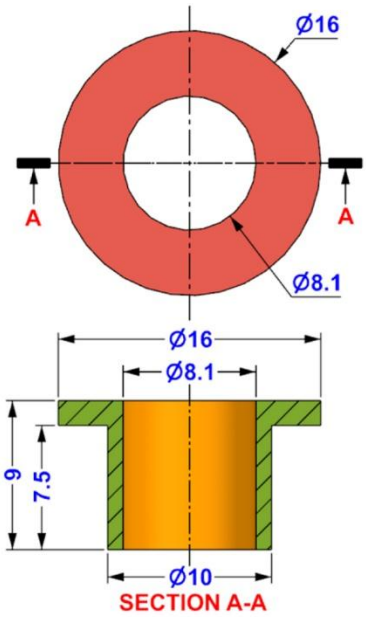
FUSION 360 ALL IN ONE WORKBOOK

500+ PRACTICE EXERCISES

- Sketching
- 3D Modeling
- Assembly



2D Sketching



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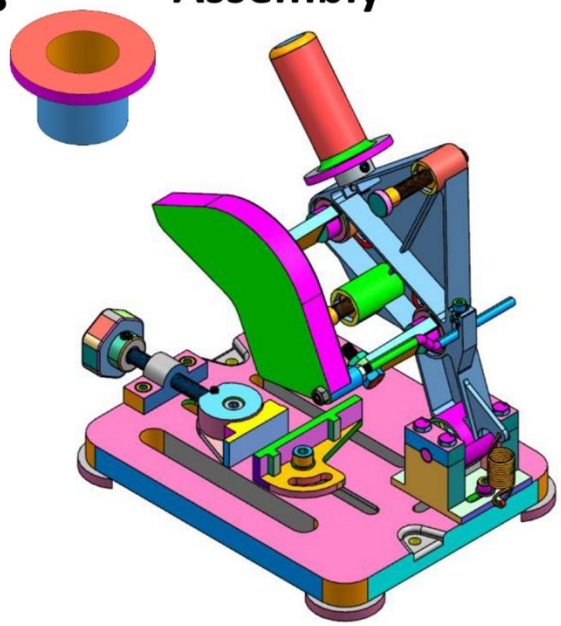

Assembly

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

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

MASTER FUSION 360: UNLOCK SKETCH MODE WITH EXPERT TIPS

• LEARN • • APPLY • • GROW •

Introduction

Entering Sketch Mode in Fusion 360 is a fundamental skill for users of this powerful 3D CAD software. As a beginner, navigating the interface and understanding how to create and work with sketches can be overwhelming. In this comprehensive guide, we'll walk you through the process of entering Sketch Mode, highlighting the key steps and best practices for creating accurate and efficient sketches.

Preparing for Sketch Mode

Before entering Sketch Mode, it's essential to understand the fundamental concepts of Fusion 360 and its interface. Familiarize yourself with the user interface, including the toolbar, menus, and workspace. Make sure you have a clear understanding of the different modes in Fusion 360, such as Part, Assembly, and Sketch Mode.

Creating a New Sketch

To enter Sketch Mode, you need to create a new sketch. You can do this by going to the **Create** tab in the toolbar and selecting **Sketch**. Alternatively, you can use the **Sketch** tool in the **Part** tab. When creating a new sketch, you can choose from various sketch planes, such as the **XY Plane**, **XZ Plane**, or **YZ Plane**. You can also create a custom sketch plane by selecting the **Create Plane** tool.

Selecting a Sketch Plane

When selecting a sketch plane, consider the orientation and position of the plane relative to your part or assembly. For example, if you're creating a part with a cylindrical shape, it's best to create a sketch on the **XY Plane**. To select a sketch plane, click on the desired plane in the browser or use the **Plane** tool in the **Sketch** tab.

Understanding Sketch Entities

In Sketch Mode, you'll work with various entities, including lines, curves, arcs, and splines. These entities are the building blocks of your sketch, and understanding how to create and manipulate them is crucial for creating accurate and efficient sketches. In the next section, we'll delve deeper into the world of sketch entities.

Working with Sketch Entities

Sketch entities are the foundation of any sketch in Fusion 360. Understanding how to create and manipulate these entities is essential for creating accurate and efficient sketches.

Creating Lines and Curves

Lines and curves are the most basic sketch entities. You can create lines and curves using the **Line** and **Curve** tools in the **Sketch** tab. To create a line, select the **Line** tool and click on two points in the sketch plane. To create a curve, select the **Curve** tool and click on multiple points in the sketch plane.

Working with Arcs and Splines

Arcs and splines are more advanced sketch entities that can be used to create complex shapes. You can create arcs using the **Arc** tool in the **Sketch** tab. To create an arc, select the **Arc** tool and click on two points in the sketch plane. Splines are created using the **Spline** tool. To create a spline, select the **Spline** tool and click on multiple points in the sketch plane.

Understanding Constraints and Dimensions

Constraints and dimensions are essential for creating accurate and efficient sketches. Constraints define the relationships between sketch entities, while dimensions define the size and shape of the entities. In the next section, we'll explore the world of constraints and dimensions.

Understanding Constraints and Dimensions

Constraints and dimensions are crucial for creating accurate and efficient sketches. Understanding how to apply constraints and dimensions is essential for achieving the desired shape and size of your part or assembly.

Applying Constraints

Constraints define the relationships between sketch entities. You can apply various constraints, including **Coincidence**, **Perpendicular**, and **Tangent**. To apply a constraint, select two or more sketch entities and click on the desired constraint in the **Constraints** panel.

Understanding Dimensions

Dimensions define the size and shape of sketch entities. You can add various dimensions, including **Length**, **Width**, and **Angle**. To add a dimension, select a sketch entity and click on the desired dimension in the **Dimensions** panel.

Understanding Assembly Constraints

Assembly constraints define the relationships between parts in an assembly. You can apply various assembly constraints, including **Mate** and **Joint**. To apply an assembly constraint, select two or more parts in the assembly and click on the desired constraint in the **Constraints** panel.

Conclusion

Entering Sketch Mode in Fusion 360 is a fundamental skill for users of this powerful 3D CAD software. By following the steps outlined in this comprehensive guide, you'll be able to create accurate and efficient sketches with ease. Remember to always practice and experiment with different sketch entities, constraints, and dimensions to master the art of sketching in Fusion 360.

FAQ

Q: What is Sketch Mode in Fusion 360?

A: Sketch Mode is a fundamental mode in Fusion 360 where you can create and work with sketches. Sketches are two-dimensional representations of a part or assembly.

Q: How do I enter Sketch Mode in Fusion 360?

A: To enter Sketch Mode, go to the **Create** tab in the toolbar and select **Sketch**. Alternatively, you can use the **Sketch** tool in the **Part** tab.

Q: What are sketch entities?

A: Sketch entities are the building blocks of a sketch in Fusion 360. They include lines, curves, arcs, and splines.

Q: How do I create a new sketch in Fusion 360?

A: To create a new sketch, go to the **Create** tab in the toolbar and select **Sketch**. Alternatively, you can use the **Sketch** tool in the **Part** tab.

Q: What are constraints in Fusion 360?

A: Constraints define the relationships between sketch entities. You can apply various constraints, including **Coincidence**, **Perpendicular**, and **Tangent**.

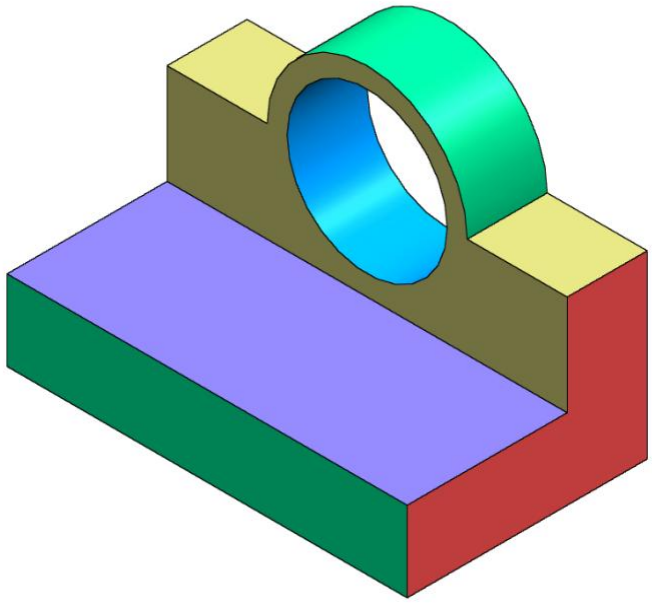
Q: What are dimensions in Fusion 360?

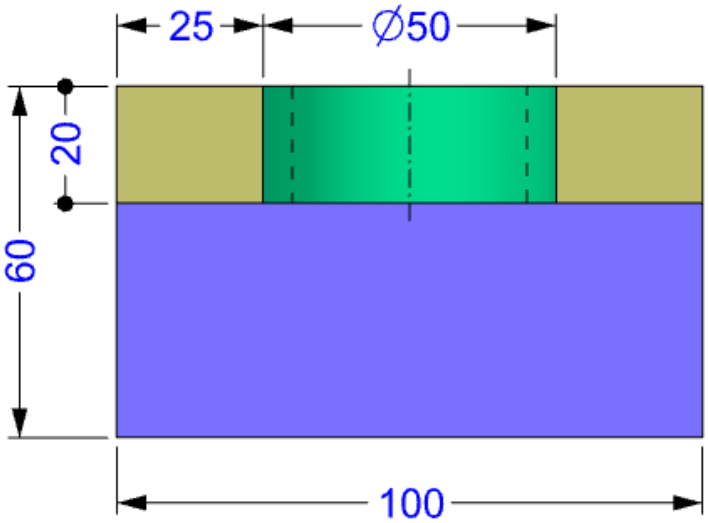
A: Dimensions define the size and shape of sketch entities. You can add various dimensions, including **Length**, **Width**, and **Angle**.

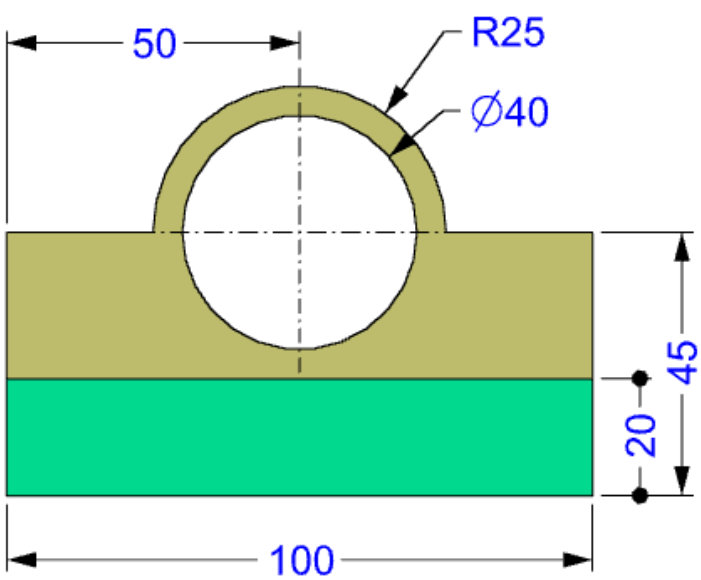
Q: How do I save a sketch in Fusion 360?

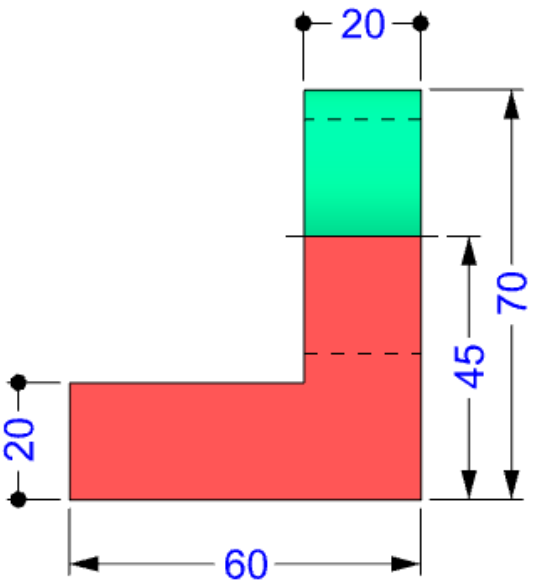
A: To save a sketch, go to the **File** menu and select **Save**. You can also use the **Ctrl+S** shortcut to save the sketch.

EXERCISE-03









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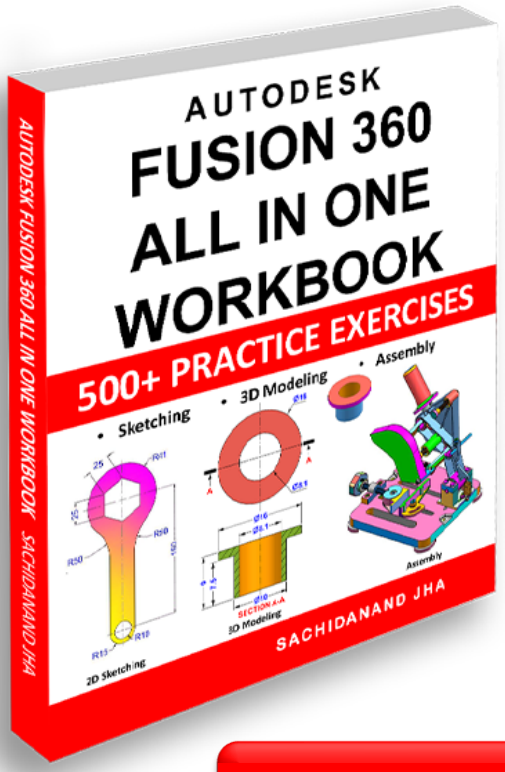

What's Included in the FUSION 360 ALL IN ONE WORKBOOK?

- ✓ 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
- ✓ 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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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)