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FUSION 360
ALL IN ONE
WORKBOOK

500+ PRACTICE EXERCISES

- Sketching
- 3D Modeling
- Assembly
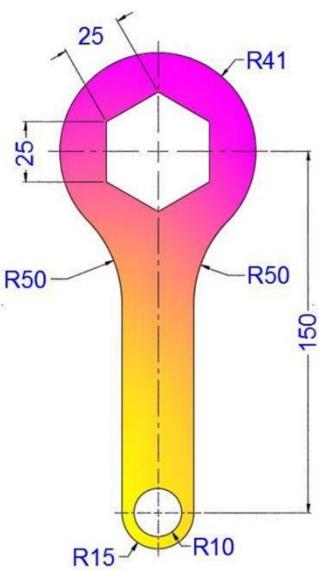
2D Sketching
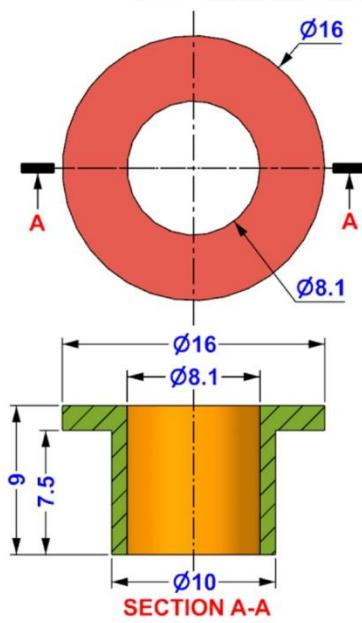
3D Modeling
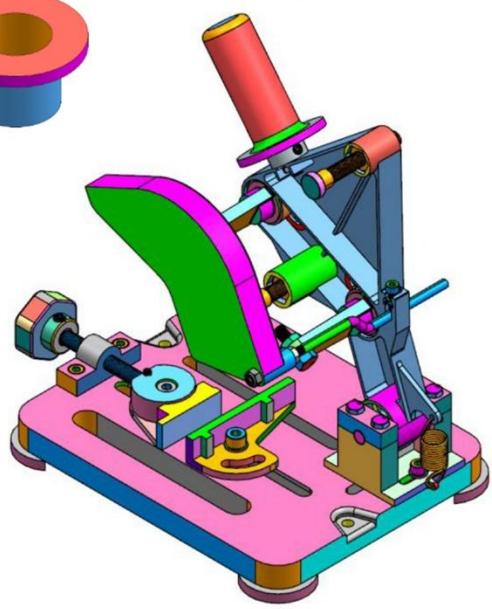
Assembly

SACHIDANAND JHA

AUTODESK

FUSION 360 ALL IN ONE WORKBOOK

500+ PRACTICE EXERCISES

2D Sketching • 3D Modeling • Assembly Drawings

SACHIDANAND JHA

cadin360°
Learning Tutorials

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

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

FUSION 360 FOR BEGINNERS: MASTERING SKETCHING ESSENTIALS

• LEARN • • APPLY • • GROW •

Introduction

Fusion 360 is a powerful computer-aided design (CAD) software that allows users to create, edit, and simulate 3D models. One of the key features of Fusion 360 is its sketching tool, which enables users to create 2D sketches that can be used as reference for 3D models or as standalone designs. In this blog post, we will cover the basics of sketching in Fusion 360, including its purpose, how to create a sketch, and some practical examples.

What is Sketching in Fusion 360?

Sketching in Fusion 360 involves creating 2D drawings or designs using a variety of tools and functions. These sketches can be used as the foundation for 3D models, or they can be used as standalone designs. Sketching is an essential part of the design process in Fusion 360, as it allows users to visualize and communicate their ideas before moving on to 3D modeling.

Purpose of Sketching in Fusion 360

The purpose of sketching in Fusion 360 is to create a 2D representation of a design concept. This can be useful for several reasons:

- It allows users to visualize their design concept before moving on to 3D modeling.
- It provides a reference point for 3D modeling, ensuring that the 3D model is accurate and consistent with the design concept.
- It enables users to make changes and revisions to their design concept before investing time and resources into 3D modeling.

Creating a Sketch in Fusion 360

To create a sketch in Fusion 360, follow these steps:

1. Open a new part or assembly in Fusion 360.
2. Select the "Sketch" tab in the top toolbar.
3. Choose the desired sketch plane from the "Sketch Plane" dropdown menu.
4. Use the various sketch tools to create lines, arcs, and other shapes.
5. Use the "Dimensions" tool to add dimensions to your sketch.

Basic Sketch Tools

The following are some of the basic sketch tools available in Fusion 360:

- **Line Tool**: The line tool is used to create straight lines in your sketch.
- **Arc Tool**: The arc tool is used to create curved lines in your sketch.
- **Circle Tool**: The circle tool is used to create circular shapes in your sketch.
- **Rectangle Tool**: The rectangle tool is used to create rectangular shapes in your sketch.
- **Polygon Tool**: The polygon tool is used to create polygonal shapes in your sketch.

Advanced Sketch Tools

In addition to the basic sketch tools, Fusion 360 also offers several advanced sketch tools, including:

- **Trim Tool**: The trim tool is used to remove or trim excess sketch elements.
- **Extend Tool**: The extend tool is used to extend or lengthen sketch elements.
- **Split Tool**: The split tool is used to split or divide sketch elements.
- **Join Tool**: The join tool is used to join or combine sketch elements.

Practical Examples

Here are a few practical examples of how to use the sketch tools in Fusion 360:

- **Example 1**: Create a simple rectangle using the rectangle tool.
- **Example 2**: Create a complex polygon using the polygon tool.
- **Example 3**: Create a circular shape using the circle tool.

Dimensions and Constraints

Dimensions and constraints are essential components of a sketch in Fusion 360. Dimensions are used to specify the size and shape of a sketch element, while constraints are used to control the movement and behavior of a sketch element.

Types of Dimensions

The following are some of the common types of dimensions used in Fusion 360:

- **Linear Dimension**: A linear dimension is used to specify the length of a sketch element.

- **Angular Dimension**: An angular dimension is used to specify the angle between two sketch elements.
- **Radius Dimension**: A radius dimension is used to specify the radius of a circular or curved sketch element.

Using Constraints

Constraints are used to control the movement and behavior of a sketch element. The following are some of the common types of constraints used in Fusion 360:

- **Fixed Constraint**: A fixed constraint is used to lock a sketch element in place.
- **Coincident Constraint**: A coincident constraint is used to align two or more sketch elements.
- **Parallel Constraint**: A parallel constraint is used to align two or more sketch elements parallel to each other.

Conclusion

Sketching in Fusion 360 is a powerful tool that allows users to create 2D designs and references for 3D models. By mastering the basics of sketching, including creating a sketch, using basic and advanced sketch tools, and adding dimensions and constraints, users can create accurate and reliable designs.

FAQ

What is the purpose of sketching in Fusion 360?

The purpose of sketching in Fusion 360 is to create a 2D representation of a design concept, which can be used as a reference point for 3D modeling or as a standalone design.

How do I create a sketch in Fusion 360?

To create a sketch in Fusion 360, open a new part or assembly, select the "Sketch" tab, choose the desired sketch plane, and use the various sketch tools to create lines, arcs, and other shapes.

What are the basic sketch tools in Fusion 360?

The basic sketch tools in Fusion 360 include the line tool, arc tool, circle tool, rectangle tool, and polygon tool.

What are the advanced sketch tools in Fusion 360?

The advanced sketch tools in Fusion 360 include the trim tool, extend tool, split tool, and join tool.

How do I add dimensions to my sketch in Fusion 360?

To add dimensions to your sketch in Fusion 360, use the "Dimensions" tool to specify the size and shape of your sketch elements.

What are the types of dimensions used in Fusion 360?

The types of dimensions used in Fusion 360 include linear dimensions, angular dimensions, and radius dimensions.

How do I use constraints in Fusion 360?

To use constraints in Fusion 360, apply a constraint to a sketch element to control its movement and behavior.

Can I undo my sketch in Fusion 360?

Yes, you can undo your sketch in Fusion 360 by using the "Undo" button or by using the keyboard shortcut Ctrl + Z (Windows) or Command + Z (Mac).

3D

EXERCISE-03

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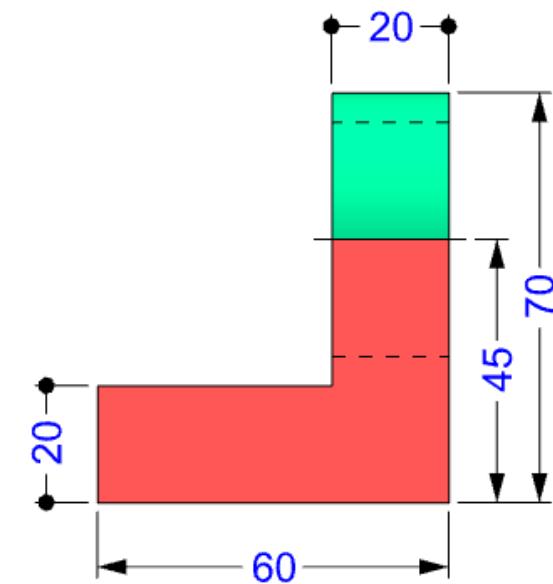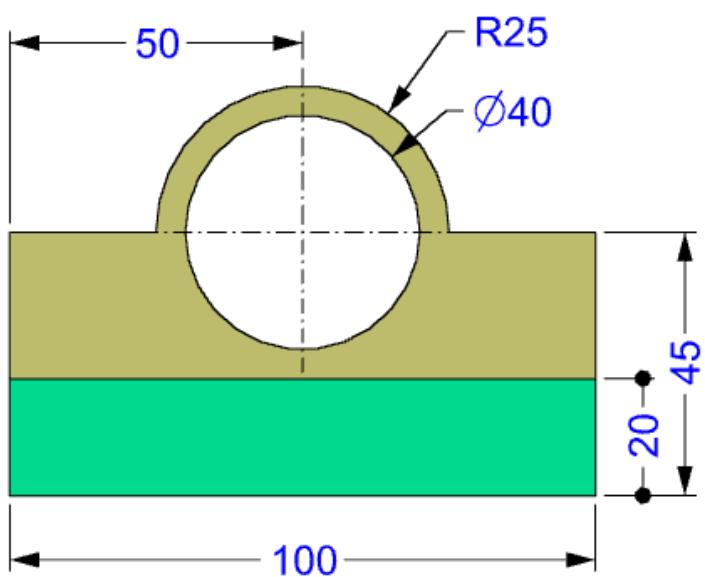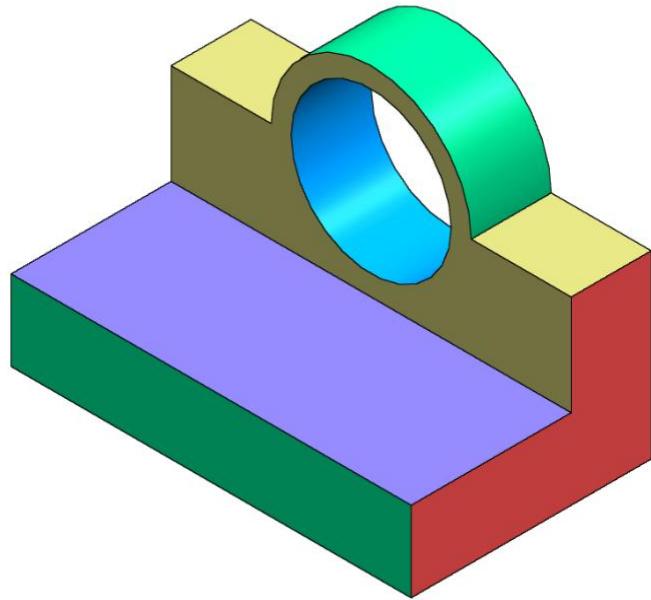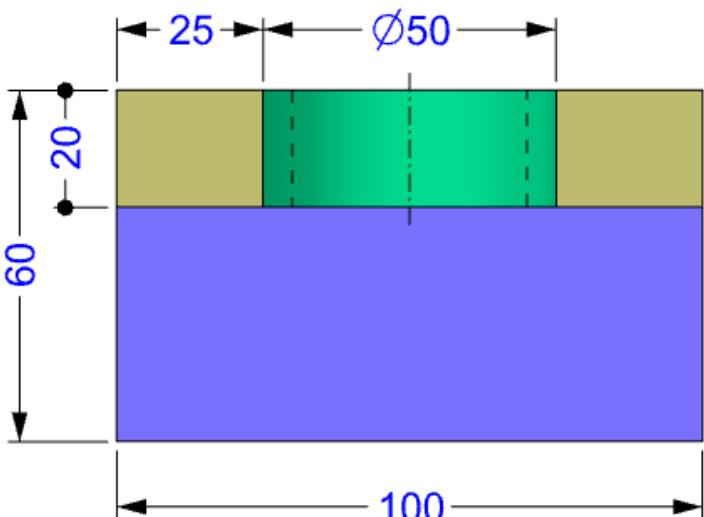
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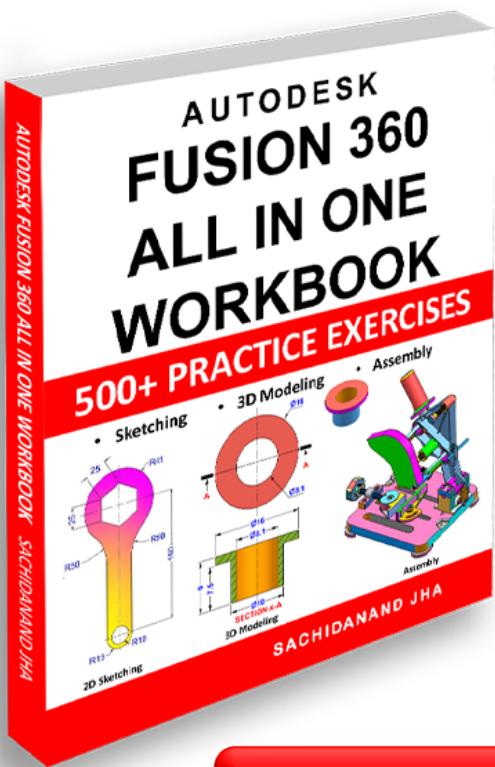
What's Included in the
FUSION 360 ALL IN ONE WORKBOOK?

- ✓ 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
- ✓ Get 200 3D Exercises in .f3d file format
- ✓ Get All Assembly Exercises in .STP file
- ✓ Instant Download Link - Sent to Your Email After Payment
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🙏 Thank You for Learning with Us!

Thank you for choosing the **AutoDesk Fusion 360 ALL IN ONE WORKBOOK**. We hope this book helped you strengthen your Fusion 360 skills through hands-on practice and real-world design challenges.

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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)