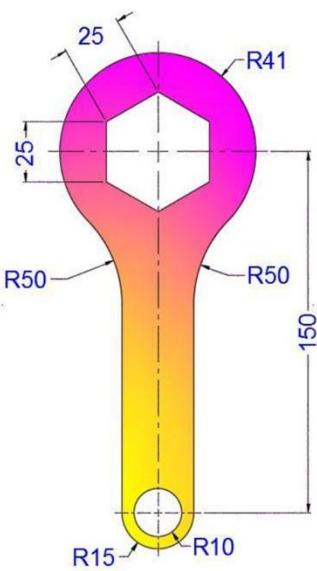


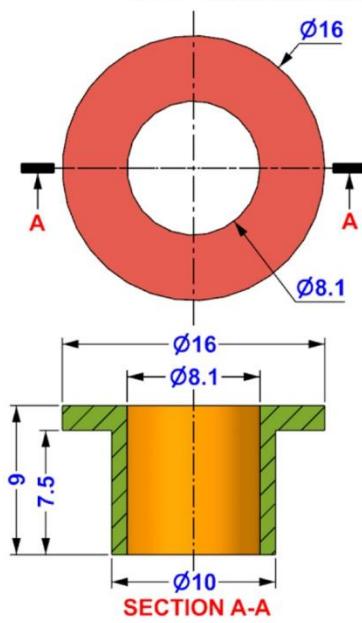
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500+ PRACTICE EXERCISES

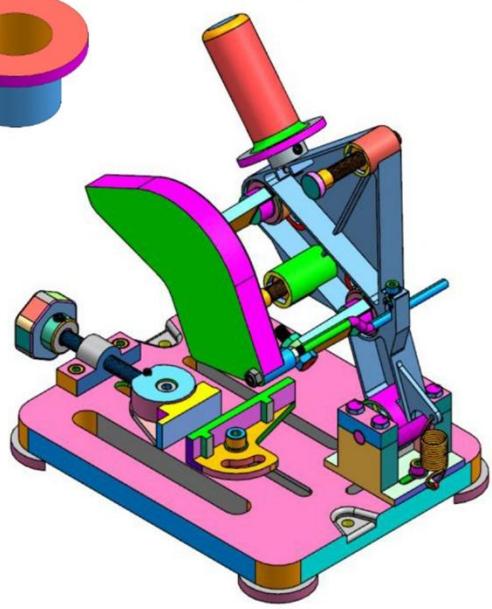
- Sketching



- 3D Modeling



- 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

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Best regards,

Sachidanand Jha
Founder & CEO, CADIN360



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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

DRAWING FACES FROM SCRATCH IN FUSION 360: A STEP-BY-STEP GUIDE

• LEARN • • APPLY • • GROW •

Introduction

Fusion 360, a powerful computer-aided design (CAD) software, offers a wide range of tools to create detailed 3D models. One of the essential tasks in 3D modeling is creating a sketch on a face, which serves as the foundation for more complex designs. In this blog post, we will guide you through the step-by-step process of starting a sketch on a face in Fusion 360.

Setting Up the Sketch Environment

Before creating a sketch, it's essential to understand the basic concepts of Fusion 360's sketching environment. Familiarize yourself with the different tools and settings that will be used throughout this tutorial.

1. **Launch Fusion 360:** Open Fusion 360 on your computer and create a new part or open an existing one.
2. **Enter Sketch Mode:** Click on the "Sketch" tab in the top navigation bar to enter the sketching environment.
3. **Select a Face:** Click on the face where you want to create the sketch. You can also use the "Select Face" tool (hotkey: F) to select the face.

Creating a New Sketch

Now that you have set up the sketch environment, it's time to create a new sketch on the selected face.

Step 1: Create a New Sketch

1. **Create a New Sketch:** Click on the "Create Sketch" button in the "Sketch" tab.
2. **Select the Face:** Make sure the face you selected earlier is still highlighted.
3. **Choose a Sketch Type:** Fusion 360 offers two types of sketches: "2D Sketch" and "3D Sketch." For this tutorial, we will use the "2D Sketch" option.

Step 2: Define the Sketch Plane

The sketch plane is the 2D plane where you will create your sketch. You can define the sketch plane manually or let Fusion 360 automatically detect it.

1. **Define the Sketch Plane:** Click on the "Define Sketch Plane" button.

2. **Select a Reference:** Choose a reference point on the face to define the sketch plane.

Adding Entities to the Sketch

Now that you have created a new sketch, it's time to add entities to it.

Step 1: Add a Point

1. **Create a Point:** Click on the "Point" tool (hotkey: P) and click on the face to create a point.
2. **Verify the Point:** Make sure the point is created on the face and is not intersecting with any other entities.

Step 2: Add a Line

1. **Create a Line:** Click on the "Line" tool (hotkey: L) and click on two points to create a line.
2. **Verify the Line:** Make sure the line is created on the face and is not intersecting with any other entities.

Modifying the Sketch Entities

As you add more entities to the sketch, you may need to modify them to achieve the desired result.

Step 1: Move an Entity

1. **Select an Entity:** Click on the entity you want to move.
2. **Move the Entity:** Click on the "Move" tool (hotkey: M) and drag the entity to its new location.

Step 2: Rotate an Entity

1. **Select an Entity:** Click on the entity you want to rotate.
2. **Rotate the Entity:** Click on the "Rotate" tool (hotkey: R) and drag the entity to its new orientation.

Conclusion

Creating a sketch on a face in Fusion 360 is a fundamental skill that requires practice and patience. By following the steps outlined in this tutorial, you should now be able to create a sketch on a face and add entities to it. Remember to experiment with different tools and settings to

achieve the desired result.

FAQ

Q: What is the difference between a 2D sketch and a 3D sketch in Fusion 360?

A: A 2D sketch is a 2D plane where you can create 2D entities, while a 3D sketch is a 3D space where you can create 3D entities.

Q: How do I select a face in Fusion 360?

A: You can select a face by clicking on it or using the "Select Face" tool (hotkey: F).

Q: What is the purpose of defining a sketch plane in Fusion 360?

A: Defining a sketch plane sets the 2D plane where you will create your sketch.

Q: Can I modify a sketch entity after it's created?

A: Yes, you can modify a sketch entity by using the "Move," "Rotate," or "Scale" tools.

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

A: You can save a sketch by clicking on the "File" menu and selecting "Save As."

Q: Can I reuse a sketch in another part or assembly in Fusion 360?

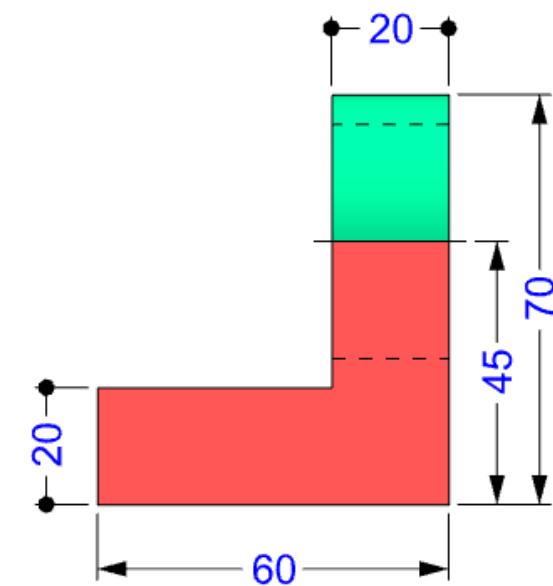
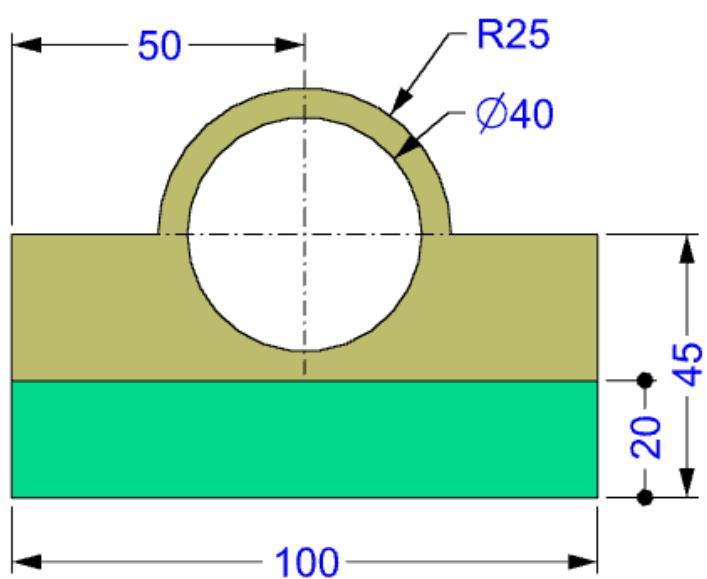
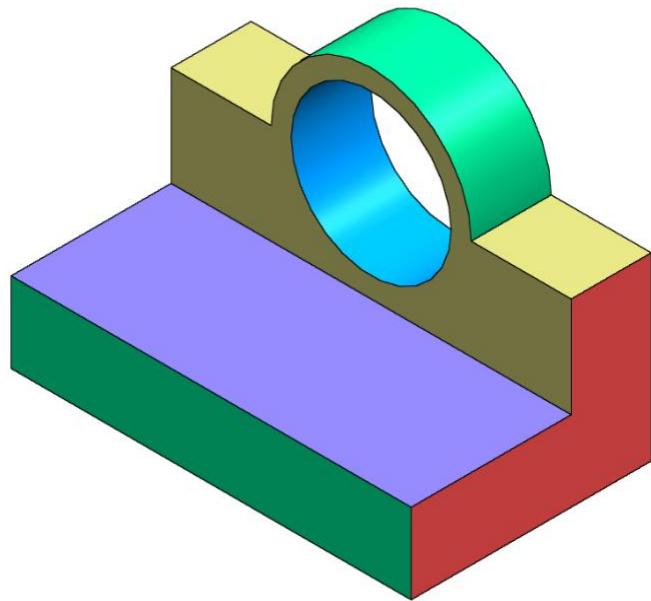
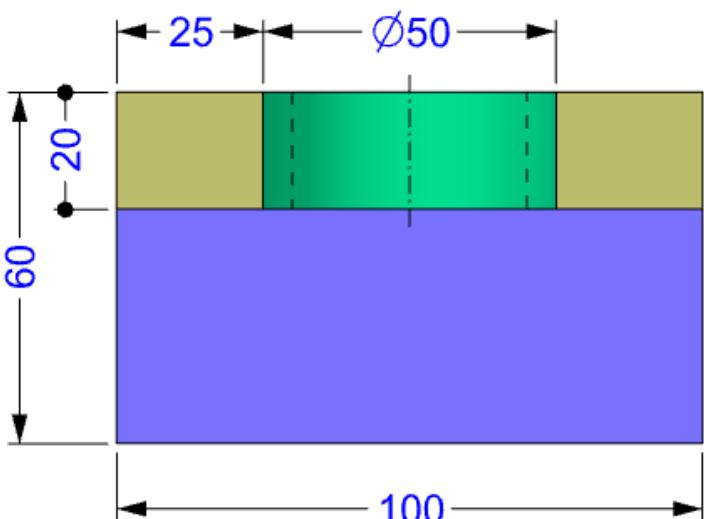
A: Yes, you can reuse a sketch by copying and pasting it into another part or assembly.

Q: What is the difference between a sketch and a feature in Fusion 360?

A: A sketch is a 2D or 3D plane where you create entities, while a feature is a 3D entity that is created from a sketch.

3D

EXERCISE-03



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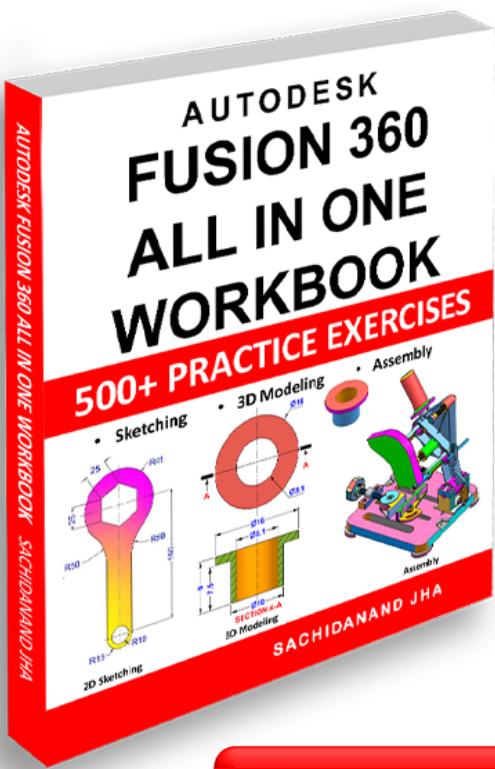
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– Team Cadin360

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- 200 3D Modeling Exercises
- Multi-part Assembly Exercises & Detailed Drawings
- All drawings in 3rd Angle projection
- All dimensions are in mm(metric system)