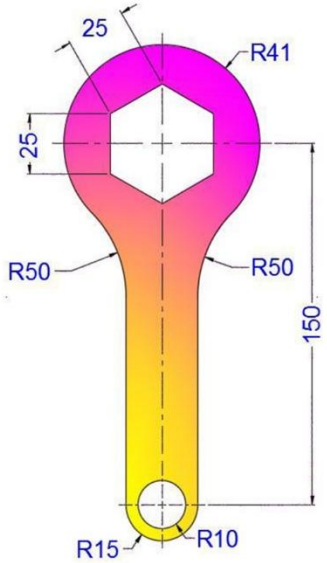
AUTODESK

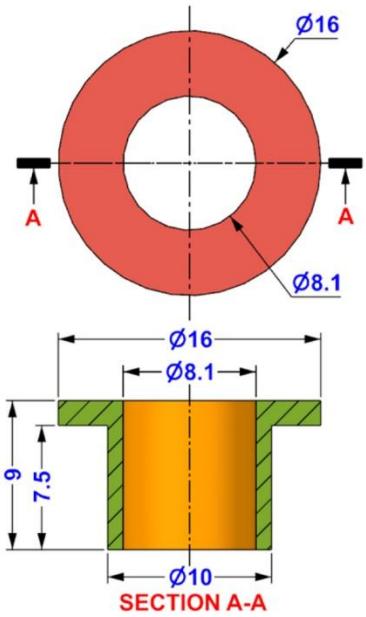
FUSION 360 ALL IN ONE WORKBOOK

500+ PRACTICE EXERCISES

- Sketching
- 3D Modeling
- Assembly



2D Sketching



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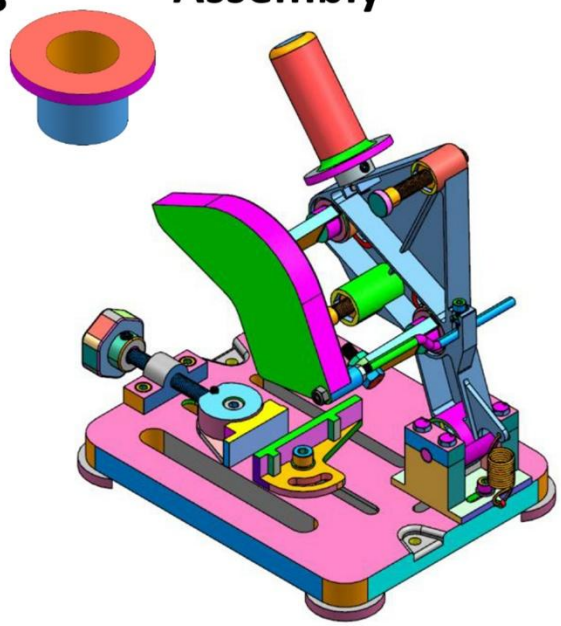

Assembly

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

SKETCH ON ORIGIN VS FACES IN FUSION 360: MASTERING THE BASICS

• LEARN • • APPLY • • GROW •

Introduction

Fusion 360, a powerful 3D computer-aided design (CAD) software, offers a variety of tools to help users create complex designs with ease. Two essential tools in Fusion 360 are sketching on origin planes and sketching on faces. Understanding when to use each of these tools is crucial for effective design creation and optimization.

Origin planes in Fusion 360 serve as a reference point for sketching and designing 3D models. They provide a stable and consistent coordinate system, allowing users to create accurate and precise sketches. On the other hand, sketching on faces enables users to create complex shapes and designs by leveraging the properties of the faces of a 3D model. In this blog post, we will explore the differences between sketching on origin planes and faces in Fusion 360 and discuss when to use each of these tools.

Understanding Origin Planes

Origin planes are the base reference points in Fusion 360 that provide a consistent and stable coordinate system for sketching and designing 3D models. There are three primary origin planes in Fusion 360: the work plane, top plane, and front plane. These planes are used as a reference point for sketching and designing, ensuring that users can create accurate and precise sketches.

Origin planes are particularly useful when creating simple and complex designs that require precise control over the coordinates. By sketching on origin planes, users can create 2D sketches that can be easily converted into 3D models. These 2D sketches can be used to create complex shapes, such as curves, arcs, and circles.

Practical Example: Sketching on the Work Plane

To demonstrate the practical application of sketching on origin planes, let's consider an example of creating a simple design on the work plane. In this example, we will create a 2D sketch of a rectangle on the work plane and then convert it into a 3D model.

1. Open Fusion 360 and create a new part.
2. Switch to the sketch workspace and select the work plane as the active plane.
3. Create a new sketch on the work plane by clicking on the "Sketch" button in the toolbar.
4. Draw a rectangle on the work plane by using the rectangle tool.
5. Convert the 2D sketch into a 3D model by using the "Extrude" tool.

Sketching on Faces

Sketching on faces is another powerful feature in Fusion 360 that allows users to create complex shapes and designs by leveraging the properties of the faces of a 3D model. When sketching on faces, users can create 2D sketches that are constrained to the surface of the face, allowing for more precise control over the design.

Sketching on faces is particularly useful when creating complex designs that require precise control over the surface properties of a 3D model. By sketching on faces, users can create 2D sketches that can be used to create complex shapes, such as curves, arcs, and circles.

Practical Example: Sketching on a Face

To demonstrate the practical application of sketching on faces, let's consider an example of creating a complex design on a face. In this example, we will create a 2D sketch on a face and then convert it into a 3D model.

1. Open Fusion 360 and create a new part.
2. Create a cylinder by using the "Cylinder" tool.
3. Select the top face of the cylinder as the active face.
4. Create a new sketch on the face by clicking on the "Sketch" button in the toolbar.
5. Draw a circle on the face by using the circle tool.
6. Convert the 2D sketch into a 3D model by using the "Extrude" tool.

Choosing Between Origin Planes and Faces

When to use origin planes versus faces in Fusion 360 depends on the specific design requirements and goals. Here are some general guidelines to help users decide:

- Use origin planes when:

+ Creating simple and complex designs that require precise control over the coordinates.

+ Creating 2D sketches that can be easily converted into 3D models.

- Use faces when:

+ Creating complex designs that require precise control over the surface properties of a 3D model.

+ Creating 2D sketches that can be used to create complex shapes, such as curves, arcs, and circles.

Conclusion

In conclusion, understanding the differences between sketching on origin planes and faces in Fusion 360 is essential for effective design creation and optimization. By knowing when to use each of these tools, users can create accurate and precise sketches that can be easily converted into 3D models. Whether creating simple or complex designs, Fusion 360's origin planes and faces provide a powerful and flexible toolset for users to unleash their creativity and innovation.

FAQ

Q: What is the difference between sketching on origin planes and sketching on faces in Fusion 360?

A: Sketching on origin planes provides a stable and consistent coordinate system for sketching and designing 3D models, while sketching on faces allows users to create complex shapes and designs by leveraging the properties of the faces of a 3D model.

Q: When should I use origin planes in Fusion 360?

A: Use origin planes when creating simple and complex designs that require precise control over the coordinates or creating 2D sketches that can be easily converted into 3D models.

Q: When should I use faces in Fusion 360?

A: Use faces when creating complex designs that require precise control over the surface properties of a 3D model or creating 2D sketches that can be used to create complex shapes, such as curves, arcs, and circles.

Q: Can I sketch on multiple faces at once in Fusion 360?

A: Yes, users can sketch on multiple faces at once in Fusion 360 by selecting multiple faces as the active faces.

Q: How do I switch between origin planes and faces in Fusion 360?

A: Users can switch between origin planes and faces in Fusion 360 by selecting the desired plane or face from the "Work Plane" or "Active Face" dropdown menus.

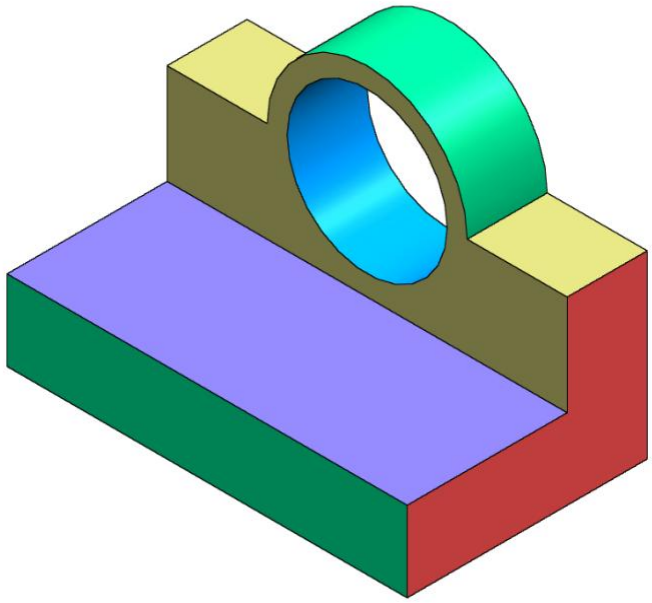
Q: Can I use origin planes and faces together in Fusion 360?

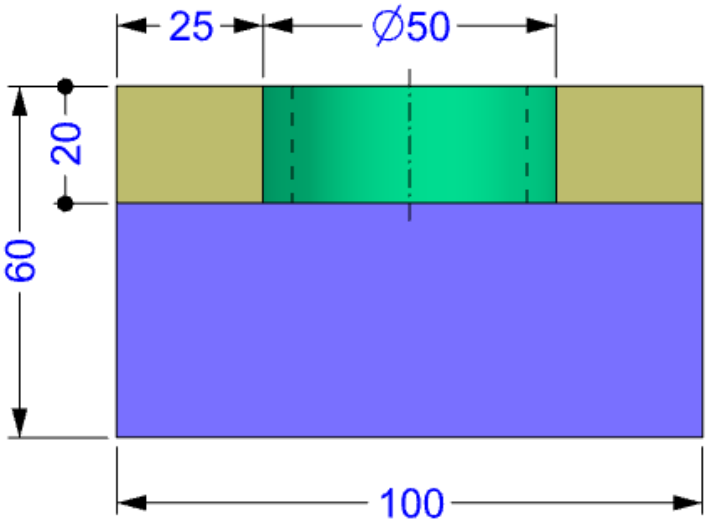
A: Yes, users can use origin planes and faces together in Fusion 360 by combining sketches created on origin planes and faces to create complex designs.

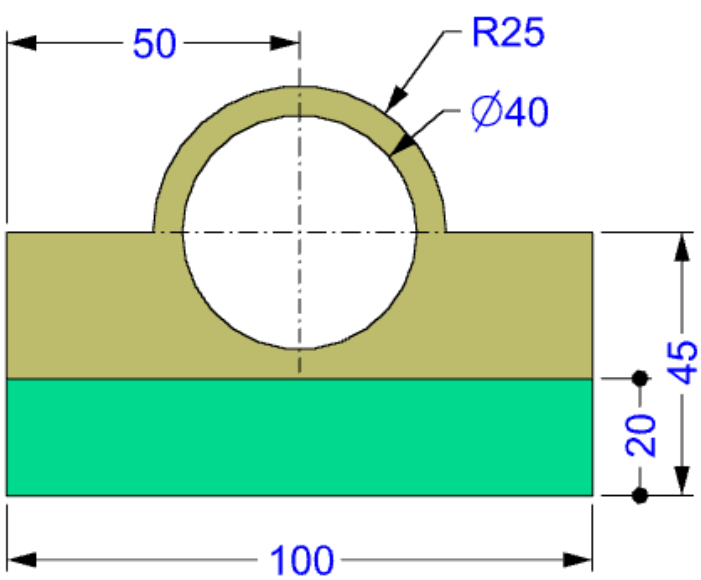
Q: What are the benefits of sketching on origin planes and faces in Fusion 360?

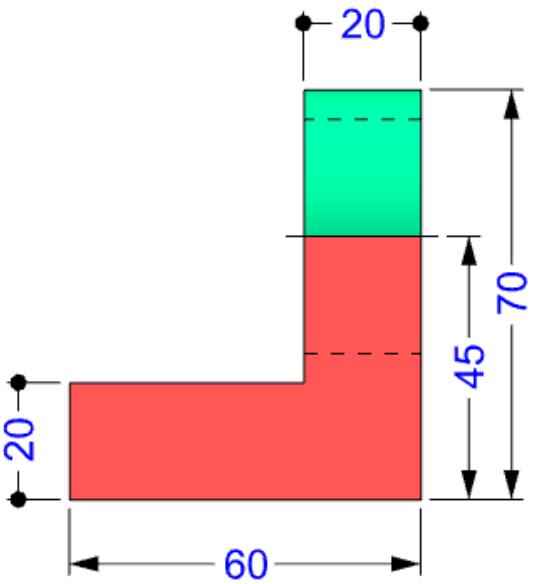
A: The benefits of sketching on origin planes and faces in Fusion 360 include precise control over design coordinates, accurate and precise sketches, and the ability to create complex shapes and designs.

EXERCISE-03









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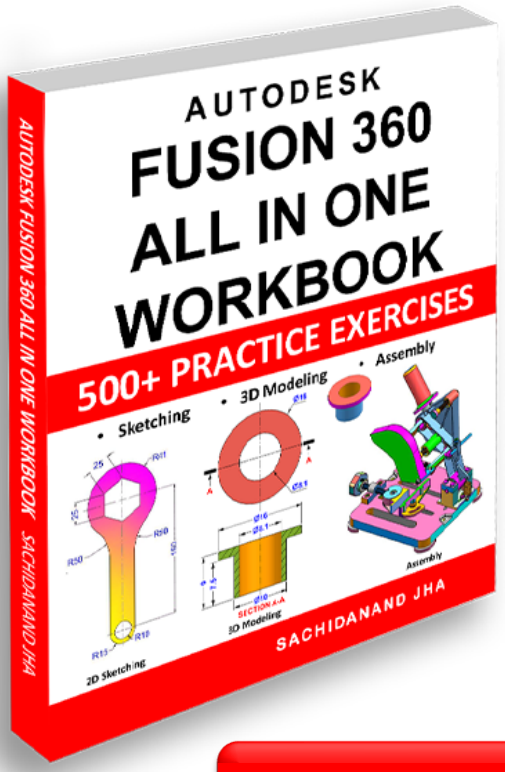

What's Included in the FUSION 360 ALL IN ONE WORKBOOK?

- ✓ 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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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)