AUTODESK FUSION 360

2026

BLOG

A Note to Our Readers

HOW TO PUSH A FACE OUTWARD IN FUSION 360

• LEARN •   • APPLY •   • GROW •

Introduction

Pushing a face outward in Fusion 360 is a fundamental modeling technique that allows designers to create complex, organic shapes or enhanced features on a solid body. Whether you're working on product prototypes, artistic designs, or detailed mechanical parts, mastering how to push a face outward is essential for adding depth and dimension to your models. This guide provides clear, step-by-step instructions for pushing a face outward effectively. It covers common mistakes to avoid, practical tips, and compares similar tools to help beginners and experienced users refine their modeling skills.

Understanding the "Push/Pull" Tool in Fusion 360

Fusion 360's primary tool for pushing a face outward is the "Press Pull" feature. It's an intuitive command that combines the capabilities of the classic extrude and move functions, allowing for flexible adjustments. Unlike simple extrudes, "Press Pull" can be applied directly on existing faces to modify models seamlessly.

Why Use the Push Face Outward Technique?

- To create outward protrusions or embellishments.
- To add depth to surface features.
- To modify complex shapes without rebuilding entire parts.
- To generate organic or aesthetic design elements.

Understanding when and why to push a face outward will help in applying the correct techniques for different design needs.

How to Push a Face Outward in Fusion 360: Step-by-Step Guide

Follow these detailed steps to push a face outward in Fusion 360 with precision and control.

1. Prepare Your Model

- Open your existing Fusion 360 model or create a new one.
- Ensure the face you want to push outward is accessible and free of complex internal geometries that may obstruct the operation.

2. Enter Sketch or Direct Modeling Mode

- For pushing faces, you typically use the "Direct Modeling" workspace.
- Switch to the "Solid" tab if you aren't already in the modeling environment.

3. Select the Face to Push Outward

- Hover over the face you intend to modify.
- Click on the face to select it. You'll notice it highlights to confirm selection.

4. Activate the "Press Pull" Command

- Find the "Press Pull" tool:
- In the "Solid" tab, click on "Modify" and then select "Press Pull."
- Or, press the shortcut key "Q" to activate it directly.

5. Push the Face Outward

- With the face selected and "Press Pull" activated:
- Click and drag the arrow handle to push the face outward visually.
- Or, enter a specific distance in the dialog box for precise control.
- Use the direction arrow to reverse the pushing direction if needed.

6. Adjust the Push Parameters

- Input the exact value for the push distance if you need precision.

- Check the "Direction" options:

- "Along a vector" to push along a specified axis or edge.

- "Normal to Face" for pushing perpendicular to the face.

7. Confirm and Finish

- Review your adjustments.

- Click "OK" or press Enter to finalize the push operation.

- The face will now protrude outward based on your parameters.

Practical Examples and Use Cases

Example 1: Creating a Lip or Flange

Suppose you're designing a box and want to add a lip on the edge. Use the "Press Pull" to push the face outward, creating an outward ridge that acts as a lip for assembly.

Example 2: Adding Organic Features

For aesthetic or ergonomic features, push a face outward along a freeform surface for smooth, organic protrusions, often used in product design.

Example 3: Reinforcing Structures

Push faces outward on structural parts to add thickness or create ribs that reinforce the design.

Common Mistakes and How to Avoid Them

- **Incorrect face selection**: Ensure you select only the face you want to modify and avoid selecting internal or neighboring faces accidentally.

- **Forgetting to set the correct push distance**: Always input specific measurements for consistency, especially for manufacturing purposes.

- **Pushing faces without considering neighboring geometry**: Pushing out can cause overlaps or unintentional intersections. Use "Replace Geometry" or "Cut" features as needed.
- **Ignoring direction controls**: Ensure you choose the correct push direction, especially when working along specific axes or normals.
- **Over-pushing**: Excessive pushes might distort the model or create non-manifold geometries, so proceed gradually.

Best Practices and Pro Tips

- Use construction planes and axes for more precise control over push direction.
- Combine "Press Pull" with "Move" or "Offset" features for complex modifications.
- Always isolate the feature you want to modify by creating components or bodies for cleaner workflow.
- For symmetrical features, consider mirroring or patterning your pushed faces.
- Save iterations regularly to compare before and after states.

Comparing "Press Pull" with Other Fusion 360 Tools

Tool	Function	Use Case	Pros	Cons

Press Pull	Push or pull faces directly	Basic extrusions or protrusions	Intuitive, flexible, quick	Limited control over complex geometry
Offset Face	Create parallel faces	Thicken or shrink surfaces	Precise, controlled	Less flexible for organic shapes
Scale	Change size proportionally	Adjust overall or selected features	Uniform control	Not suitable for complex pushes

Freeform	Sculpt or deform models	Organic and artistic design	Highly flexible, natural shapes	Steeper learning curve

Using "Press Pull" for outward pushes is preferable for most straightforward modifications, but combining it with other tools enhances modeling capabilities.

Conclusion

Learning how to push a face outward in Fusion 360 not only broadens your modeling skills but also streamlines workflows for intricate or innovative designs. By mastering the "Press Pull" technique—through precise control, understanding directional options, and avoiding common pitfalls—you can confidently add depth and dimension to your models. Remember to experiment with different parameters and paths to realize your creative vision effectively.

FAQ

1. How do I push a face outward along a specific axis in Fusion 360?

Ans: Select the face, activate "Press Pull," and in the dialog box, specify the direction along your desired axis, or use the "Direction" options to align with a construction axis.

2. Can I push multiple faces outward at once?

Ans: Yes, select multiple faces simultaneously before activating "Press Pull" to push all selected faces together.

3. What should I do if the face pushes interfere with other geometry?

Ans: Consider creating separate components or bodies for complex features and use "Cut" or "Combine" operations to manage interference.

4. Is there a limit to how much I can push a face outward?

Ans: Practically, there's no strict limit, but overly large pushes can distort the model or cause geometry issues; always push gradually and review results.

5. How do I push a face outward evenly on all sides?

Ans: Use the "Offset Face" tool for equal thickness adjustments or push individual faces in a synchronized manner with consistent distances.

6. Can I animate a face being pushed outward?

Ans: Fusion 360 doesn't directly support animation of pushes, but parameters can be linked with timelines or joint drives in Fusion 360 with the "Parameters" feature.

7. What's the best way to learn more advanced pushing techniques?

Ans: Experiment with combining "Press Pull" with sculpting tools, and consult Fusion 360 tutorials focused on direct modeling and freeform design.

About CADIN360

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 Autodesk Fusion 360.

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

You've just completed this blog and learned important concepts in Autodesk Fusion 360.

To help you practice and apply what you've learned, the next pages include a sample from our Fusion 360 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 Fusion 360 practice starts next.

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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

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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.
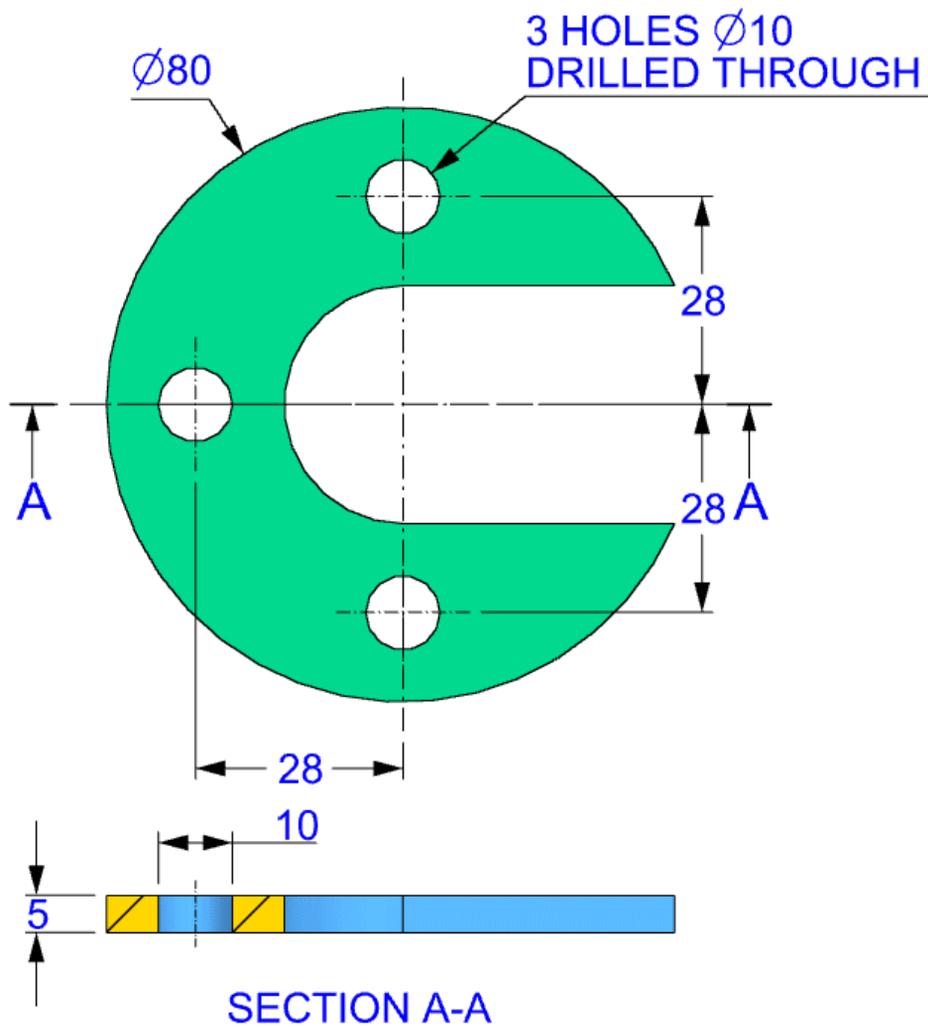- This book is ideal for both students and professionals preparing for industry design work.

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

Happy learning!
– Team CADIN360

3D

EXERCISE-01

3 HOLES ∅10
DRILLED THROUGH

∅80

28

28 A

A

28

10

5

SECTION A-A

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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)