100 Things Beginners Should Learn in Fusion 360

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Learning Objectives

Learning Objective 1
Learn sketching best practices and tips.

Learning Objective 2
Discover modeling strategies and tips.

Learning Objective 3
Learn about imports and file management.

Learning Objective 4
Learn strategies for assemblies, 3D prints, and more.

Description

Do you want to cover 100 concepts that all beginners seem to need and ask? This class is the answer to all those questions that students have asked the speaker throughout the years in classes and on their YouTube channel. We will cover high-level strategies, small tips and tricks, ideal settings, and other resources you should know. Learn proper sketching strategies (easier to design and make changes—save time now and when you need to edit). Learn better modeling—learn the techniques for tackling simple to complex designs, and feel more confident when you start up Fusion 360 software. Learn all the cool resources available to beginner users, and design and 3D print what you want.

Speaker(s)

He has over 15 years of experience in the design and manufacturing space and has learned from industry experts at companies like Autodesk and SOLIDWORKS. He’s passionate about solving problems for customers that lets them achieve their bigger goals.
Tyler regularly posts Fusion 360 content on youtube @ https://bit.ly/SubscribeTechEspressoNow
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1  CORE TIPS

1.1  Start with Save Command

1.2  Recovery Documents and Settings

1.3  S Key To Search and Favorites

1.4  Familiarize Yourself with Menus
Take time to learn the many different capabilities in Fusion 360.
1.5 Google if you know the feature in another tool like Solidworks

1.6 Escape Key
Finishes out commands and let’s you start fresh with new ones.

1.7 X on Menus

1.8 Closing A Sketch And How To Start Editing From Timeline

1.9 Magic Right Click

1.10 3 Button Mouse
Consider purchasing an inexpensive 3 button mouse if you don't already have one.
1.11 Navigation Cube Is Your Friend

1.12 Hollow Quick With Shell

1.13 Investigate with Section Analysis

1.14 The Timeline

1.15 Primitives

Quick and dirty way to create objects in Fusion 360
2 Better Modeling

2.1 Plan Out Design (Remove Complexity)

2.2 Work Backwards
Remove the finishing touches (fillets and chamfers) and cuts. Think about the solids.

2.3 Use Extrudes and Revolves
Whenever possible, these two features should be your go to.

2.4 Solids, Cuts and Finishing Touches

2.5 Do Things In Chunks
Don't be afraid to add multiple extrudes. Don't try to model everything in one step.
3 SKETCHING & RELATED TIPS

3.1 CREATE SIMPLE SKETCHES
More sketches and more features is a better performing modeling practice.

3.2 Too many entities leads to poor performance and editing
Sketches with lots of entities can become hard to edit.

3.3 Use front, top, right plane often
Using standard planes provides more control and efficiency.

3.4 Keep origin in the center (when possible)
Origin in the center of a part means there will be 3 planes.

3.5 Dimension or relate to origin
To fully define a sketch you must relate or dimension to the origin.

3.6 Fully define your sketch for design intent
Fully defined sketches make for more control of a sketch and protect against unwanted edits.

3.7 Constraints then dimensions
General rule of thumb. Constraints before dimensions leads to better control of you design.

3.8 Drag blue entities to find what still needs a constraint
When unsure of what is missing, dragging blue entities exposes what still needs a constraint or dimension.

3.9 Sketch has issue – draw lines across it
Find your errors with the line method.
3.10 One extra dimension to fully defined sketch over constrains
When fully defined. No more dimensions or constraints can be added. You must remove something to add something.

3.11 Delete unnecessary dimension
Remove a dimension preventing the change you want to make.

3.12 Change from driving dimensions to a reference dimension (driven)
Driven dimensions are references or measurements that are passive. They cannot be double clicked and changed.

3.13 Name sketches
Provide names to make future edits easier.

3.14 Reuse sketches (when simple enough)
With simpler designs you can reuse sketches for faster modeling and edits. Large complex designs can make this harder.
3.15 Select tool (select other)
3.16 For multiple depths use multiple extrudes

3.17 Trim isn’t needed

3.18 Extrude sub regions

3.19 S key (search)

3.20 Break (break up sketch entities)
3.21 Parameters (equations)

Drive multiple dimensions and use any formulas.

3.22 List all dimensions (parameters tool)

3.23 Fx next to a dimension (driven by equation)

3.24 S key (sensitive to which tool you are using sketches, features)
One of my favorite tools for searching and for finding my favorite commands.

3.25 Add faves to s key shortcut toolbar

3.26 Use rectangles as often as possible
Rectangles have built in constraints and quick dimensions.
3.27 Hit tab when sketching rectangles

3.28 Sketch grid
Optional if you like this method.

3.29 Grid settings and snaps
Sketching with snapping to a grid is an option. Some users prefer this.

3.30 Palette constraints
3.31 Look at
Easy to look directly at a face or plane.

3.32 Slice tool
Section viewing tool for sketches.

3.33 Turn off/on dimensions
If you need to see your sketch with and without dimensions.

3.34 Turn off/on constraints
Easy to see your sketch with and without constraints.

3.35 Change units
How to switch between units when needed.

3.36 Set default units
Do you have a preferred units to start in your designs? Set that default.

3.37 Trim isn’t needed
With regions and break, you don’t have to trim.

3.38 CLICK THEN CLICK TO CHAIN WHEN SKETCHING
For sketching connected lines.

3.39 Hit Escape To Stop

3.40 WINDOW SELECT TO LEFT VS SELECT TO THE RIGHT- Q
Selection dragging towards the left will grab anything box touches or includes.
3.41 DIMENSION TO MIDPOINTS (HOLD SHIFT)

3.42 QUICK MEASURE (SELECT A LINE)
Find measurements quickly by selecting without having to use the measure tool.

3.43 HOLD CONTROL TO SELECT MULTIPLES
Command on a Mac.

3.44 SLOT TOOL

3.45 DIMENSION ARC TO ARC

3.46 REUSE GEOMETRY

3.47 RECREATE SKETCH LINES FROM EDGES

3.48 OFFSET ZERO

3.49 LINE TO ARC SHORTCUT

3.50 AVOID SNAPPING (HOLD CONTROL)
4 IMPORT AND EDIT

4.1 FILE UPLOAD (CLOUD TRANSLATION)

Cloud Translator support these files.

4.2 Toggle design history setting
Right click on the file name

4.3 DELETE SELECTED FACES (PATCH AND REFILL HAPPENS)
4.4 Select fillets and hit delete – add another fillet
4.5 Select faces of rib and back face
4.6 Select faces to stretch (hit q)
4.7 Pull faces or delete depending on outcome

4.8 Select faces and translate

4.9 Select faces and rotate about an axis
5 DRAWINGS

5.1 Save file
Need to save to get started.

5.2 Pick standard (iso/ansi)

5.3 Units (1 chance)
Unfortunately you cannot change the units on the 2D drawing after you select.

5.4 Duplicate units possible
You can show both units on a dimension.
5.5 Drawing size

5.6 Base view first

5.7 Projecting views
Create intelligent views quickly.

5.8 Click continue
Drag and drop next view then click the check mark.

5.9 Adjust edge visibility
For rounded edges in a drawing view.
5.10 Tab for drawing
Drawing is on a different tab than the part or assembly design.

5.11 Drawing has own version
Design files have saved versions separate of the drawing versions.

5.12 3rd angle projection
1st and 3rd angle projection are the two options. 3rd angle is widely used in North America.
5.13 Section views & snapping to
To align the section view cutting tool you can snap to different entities.
5.14 Detail views
Uses circle tool to do a super zoom on the drawing view.

5.15 Automatic center patterns & center marks
Quick tool for picking up patterns.
5.16 Reference dimensions
Dimensions in the drawing more like measurements or references.

5.17 Text/notes

5.18 Gd&T, surface textures, title block

5.19 Update model and update drawing

5.20 Export Types (PDF)
With a regular license of Fusion you have many export options. (Free license has less.)
6 SETTINGS

6.1 Auto Project Edges

6.2 Auto Look At Sketch

6.3 Edit Dimension When Created

6.4 Auto Project Geometry

6.5 Scale Entire Sketch At First Dimension
7 3D PRINTING

7.1 Export to preferred slicer

7.2 Manufacture space

7.3 Basic slicer but you can stay in fusion
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4.4 Keep origin in the center (when possible)

4.5 Dimension or relate to origin

4.6 Fully define your sketch for design intent

4.7 Constraints then dimensions

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6.1.1 Top 10 tips for Fusion 360 beginners!

https://medium.com/@FusePS/top-10-tips-for-fusion-360-beginners-8a26f7040b52
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