PD18157

Fusion 360 101: Introducing Fusion 360

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Learning Objectives

- Getting Fusion 360 installed and log in
- Learn how to create a simple sketch and part
- Learn how to make a simple assembly relationship
- How to render and/or print your design

Description

Heard about Fusion 360 software? Seen a few videos? Haven't taken the plunge yet? Online tutorials not your thing? This class is for people who want an in-person and up-close introduction to Fusion 360 software. We'll cover the basics, from login to exit, sketch to part, to assembly and far beyond. The intent is to give you an overview that will help you understand the scope and power of Fusion 360 software, whether you're a CAD veteran or just curious about what it can do for you. This session features Fusion 360 and Fusion Team.

Your AU Expert(s)

Phil Eichmiller has worked with Inventor software as a product designer for 16 years, and he's been teaching Inventor software for the CAD Program at Portland Community College for the last 8 years. He currently is also a senior software quality assurance engineer for Autodesk, Inc., on the Fusion 360 Team, where he enjoys testing, presenting, and teaching about Fusion 360 software. Roller derby is his favorite pastime, especially watching his daughters, who are both junior roller derby stars in Portland, Oregon.

Aaron Magnin attended the University of Nevada, Reno, where he obtained a degree in Mechanical Engineering. After a number of years in industry, he found his way into application engineering where he focused on simulation products, technical presentations, training, and technical support. He is a technical marketing manager at Autodesk, and creates many of the video you see on the YouTube channel.

Bryce Heventhal is the Technical Marketing Manager for Fusion 360. He has 9+ years of CAD and FEA experience, with a B.S. in Mechanical Engineering. He creates many of the Fusion 360 videos you see on YouTube. When he is not designing he is out rock climbing.

## Getting Fusion 360 installed and logging in

The first things you want to do are to install Fusion 360 and then create an Autodesk ID if you don't already have one. The installation is quick and easy and so is registering to get your ID. Please note that Fusion runs on both Mac and Windows operating systems. Also, you only need one Autodesk ID for all of our applications and web sites. All of your Fusion 360 design data is tied to this login so try to avoid making a new ID if you already have one.

Streaming Installation and creating your ID
The link to install is on the Fusion 360 home page. Once you click the download button you will be prompted for an email address and a new page will open showing you links to vital information for getting started. The streaming installation takes just a couple minutes, depending on your connection speed.

Mac or Windows
Fusion 360 runs natively on macOS and Win 7 through 10 (64 bit). Unlike other Windows programs that get "shoe horned" into Mac, Fusion is developed for both platforms from the ground up. Many current customers use both. Of course, this means the data you create will open and edit equally on any supported operating system.
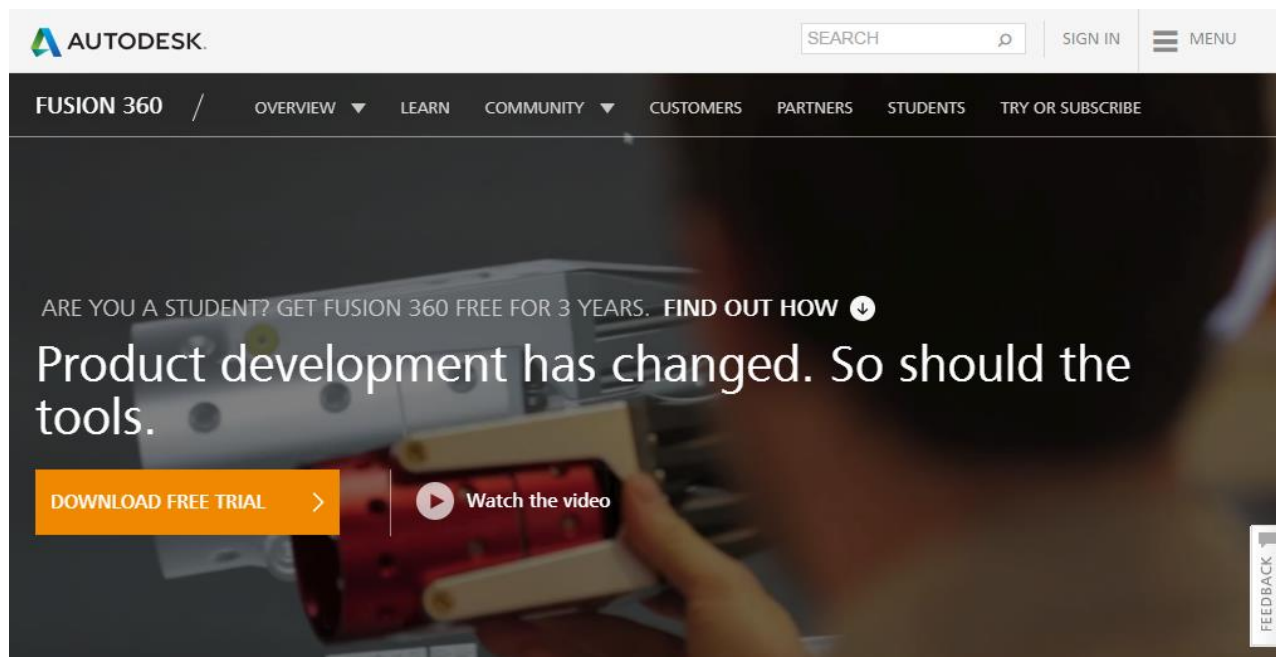
FUSION 360 HOME PAGE

If you already have an Autodesk ID
Be sure to use your existing email address associated with your existing Autodesk ID.

Try or Subscribe
All new Fusion customers are given a 30 day free trial. While on the 30 day free trial you get all the functionality of Ultimate level subscription.

After the initial 30 day trial period:
- Students can register for a free 3 year educational license (Ultimate).
- Hobbyists and business startups can register for 1 year of free trial (Ultimate).
- Subscribers will be offered base Fusion or Ultimate when they purchase.

All of the data you create while trying Fusion will always be yours for any purpose no matter your current entitlement. So jump right in and enjoy all the power and fun of Fusion 360 with no worries, it's your data.

Mac App Store
Fusion 360 is also available on the Mac App Store.

Fast track for SolidWorks experts
The getting started page has several links to help you get going. If you are a SolidWorks user there is a special Fast Track site just for you. You can also contact us directly, get system requirements, or download the security white paper.

Students and Educators
This section describes the special steps and help available for installation of Fusion 360 for students and educators in a school setting. Individual students who wish to try Fusion 360 on their own should follow the instructions above under *Streaming Installation and creating your ID*.

(See Appendix A for detailed instructions for educational institutions)
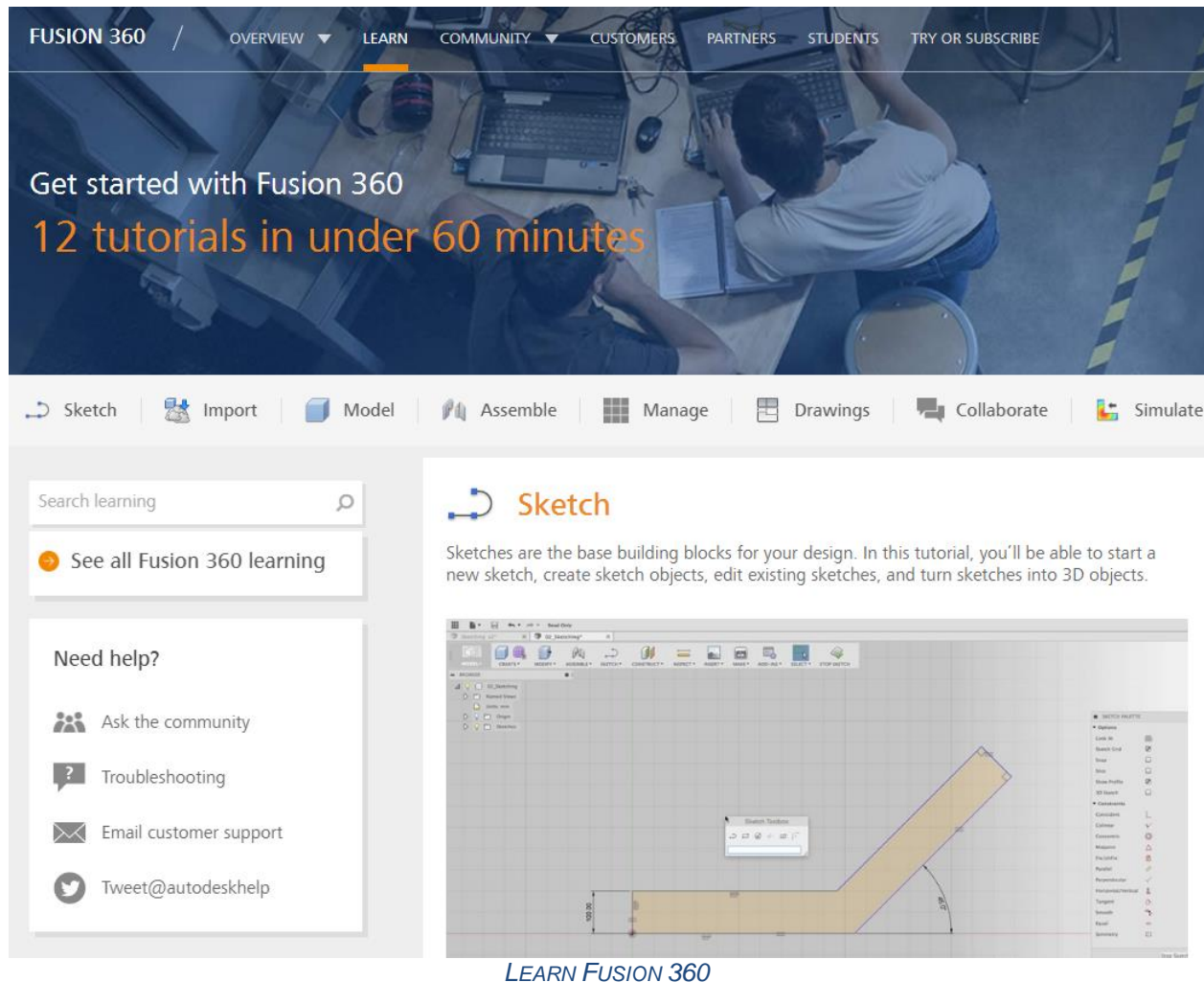
While at AU 2016
While you are at AU there are many Fusion 360 classes available for all skill levels. Don't worry if you can't attend them all, these will be recorded and available after AU. At the end of each section of this paper, there are links to related classes for that section.

Related AU 2016 class (for educational institutions):
- MP21571 - Creating a Global Classroom Where Education and Industry Collaborate Using Fusion 360

Making your first sketch and part in Fusion 360

Now that you have Fusion 360 installed it's time to learn. There are tools to help you get started such as the Fusion 360 learning site tutorials, or perhaps the Fusion 360 YouTube channel if videos are your thing.

LEARN FUSION 360

This class, Fusion 101, is an introduction to Fusion and is not comprehensive. Through the following workflow example of a simple sketch-to-part demonstration we want to introduce you to Fusion's user interface and most basic workflow. When you get down to serious learning you should be familiar with the concepts presented here.

Sketching - getting started in the Fusion modeling environment

Most mechanical parts start off with some kind of 2D sketch that gets extruded, revolved, or lofted to create a 3D shape. You can keep adding sketches and 3D shapes by repeating these steps. Or additional features can be placed without sketching, such as fillets placed by selecting edges to fillet. Fusion is no different in this regard. For this introduction we are starting in the Modeling environment where sketches and modeling commands are used to define the geometry of your design. Other environments such as CAM or SIM have access to the model you create in Modeling. It's all one data set!

The modeling interface

When Fusion opens you'll find a blank modeling environment called the canvas. The tools you need to create your first model are all visible. There are navigation tools (1) to help you get around your model. Your first project is loaded in the data panel (2), this is where your uploaded models and saved Fusion creations will be displayed, and it's your filing cabinet in the sky. The toolbar (3) has all of your 2D and 3D modeling tools.
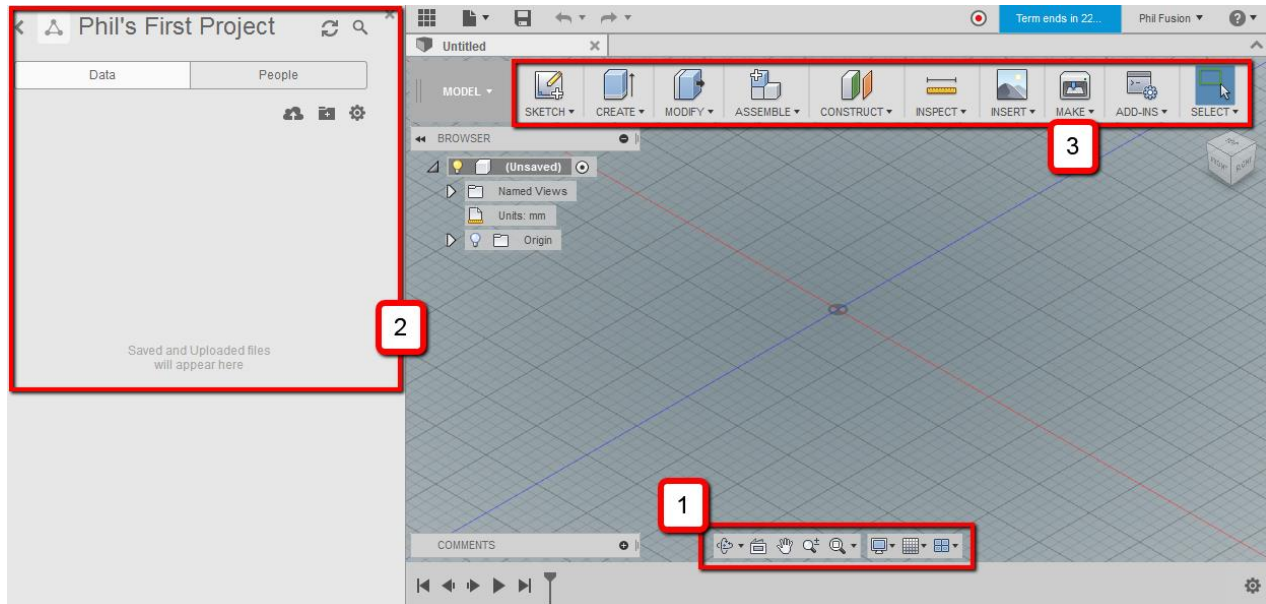
THE MODELING INTERFACE

Starting a sketch

Starting your first 3D sketch is as simple as starting a sketch command, such as line, and then picking a reference plane. In a brand new part, the reference plane is usually one of the origin planes.
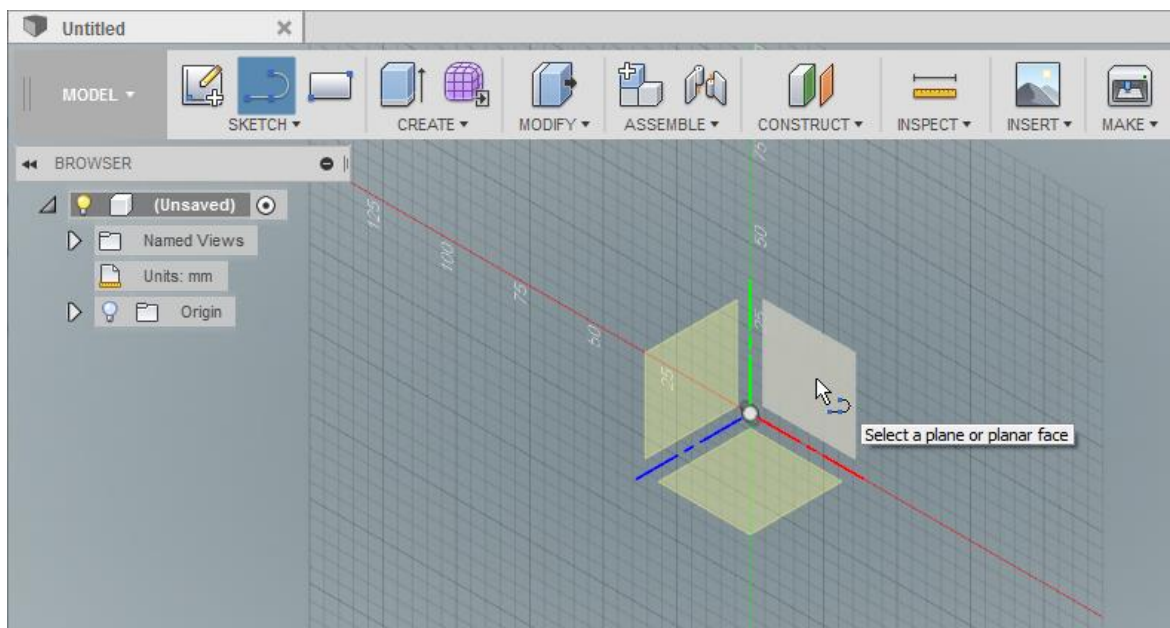
STARTING A SKETCH LINE ON THE X-Y PLANE

Adding geometry to your sketch

If you are using a sketching command such as Line you will immediately begin drawing your sketch using that tool. Additional tools are available on the Sketch dropdown menu.

You will also notice a few key features found while sketching: The browser (1) is where items you create are listed. A sketch folder has been created and your first sketch is stored for editing inside this folder. A heads-up display (2) dynamically follows the mouse action, allowing you to input numbers and parameters on the fly. The sketch palette (3) only appears when you are sketching to give you easy access to options, geometric constraints, and some context based information for various sketch workflows.
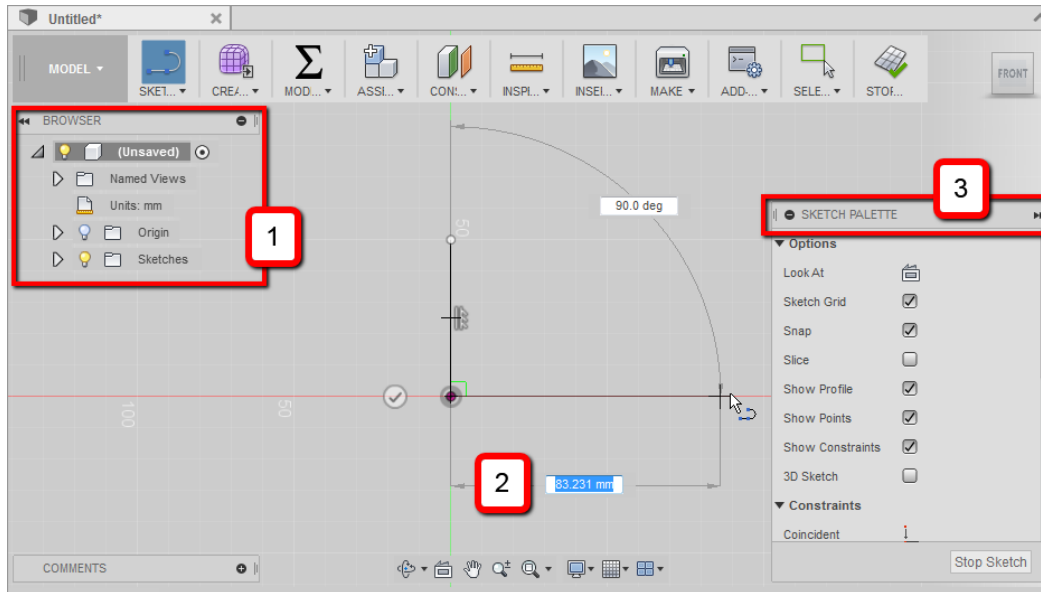
ADDING GEOMETRY TO SKETCHES

Defining size with sketch dimensions

Before your sketch is complete you should dimension any undefined sizes that define your profile. Use the dimension command in the Sketch menu to place dimensions on sketch objects. There is only one dimension command and it operates based on context. If you pick a line, you'll get an automatically aligned linear dimension. If you pick an arc, you get a radius dimension.
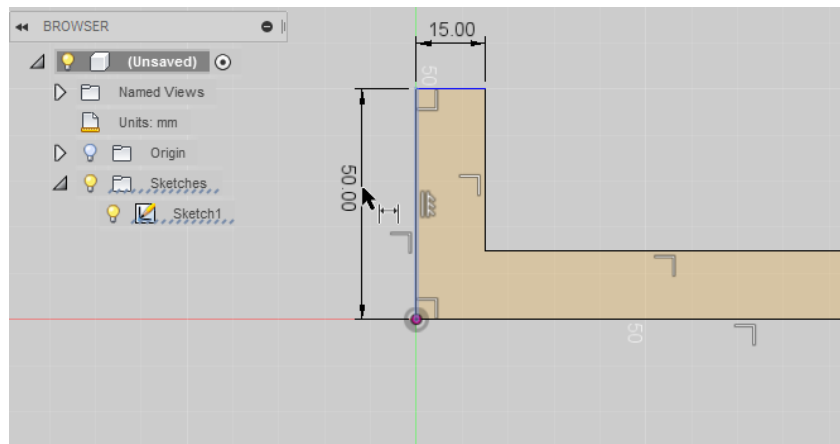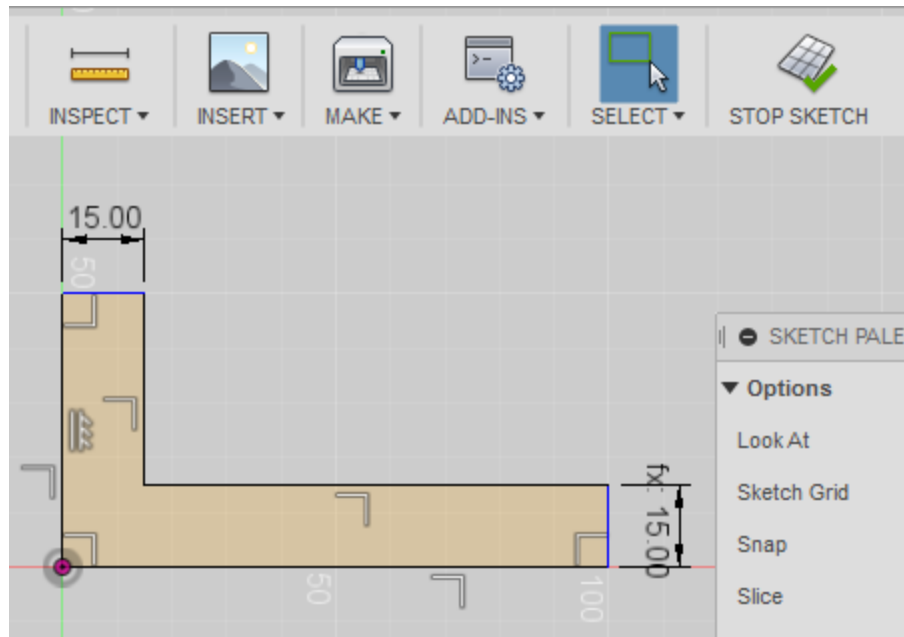

USING SKETCH DIMENSIONS TO DEFINE SIZE

Stop the sketches – Or Not!

Fusion 360 has a stop sketch command. When you are done sketching, or editing a sketch, use this command to formally end the sketch and begin 3D modeling commands.

STOP SKETCH BUTTON

TIP:

By simply starting the extrude command, or any other Fusion command, the sketch will finish automatically and the next command will begin. This is true almost everywhere you go in Fusion 360. Starting the next command will "commit" the previous one, which cuts down on clicks and trips to the dialog box just to say OK.

Make it 3D

However you want to end the sketch, you are now ready to use a 3D modeling command such as Extrude or Revolve. The example below is for extruding the L shape to make an awesome bracket.

In the picture (next page) you see the extrude dialog (1) with common modeling options.

Profile: This is the yellow area inside the boundary created by the sketched lines. It's what you are extruding.

Distance: Use this field or the HUD (2) to enter a distance. The blue arrow (3) can also be dragged. The dropdown menu here also has recently used values and a measure tool for picking the distance on screen. Lots of options here!

Taper angle: Enter a value or use the manipulator for it, see (3).

Direction: Has option for either direction, Symmetric or Two Side (with different values).

Operation: In this example we are creating a single new body in a single part file. You can also pick Boolean operations such as cut.

Extents: Use a distance or pick a condition such as To Object.
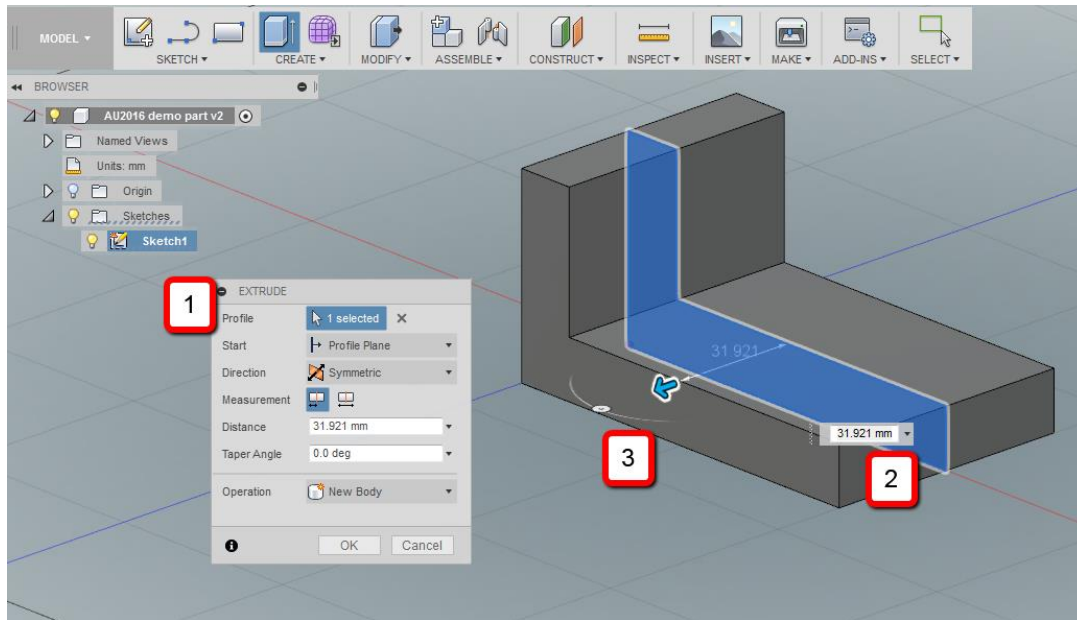
EXTRUDING THE FIRST SKETCH TO MAKE YOUR FIRST BODY

Adding sketched features

Once you have a basic body defined in 3D you can add more features using additional sketches. Just pick a sketch tool (1) and pick a plane to draw on (2). You can also start to see your actions recorded in the timeline (3) which is your tool for managing "when" things happen in your model. For example: moving the timeline marker to add sketches and features *prior* to other actions.
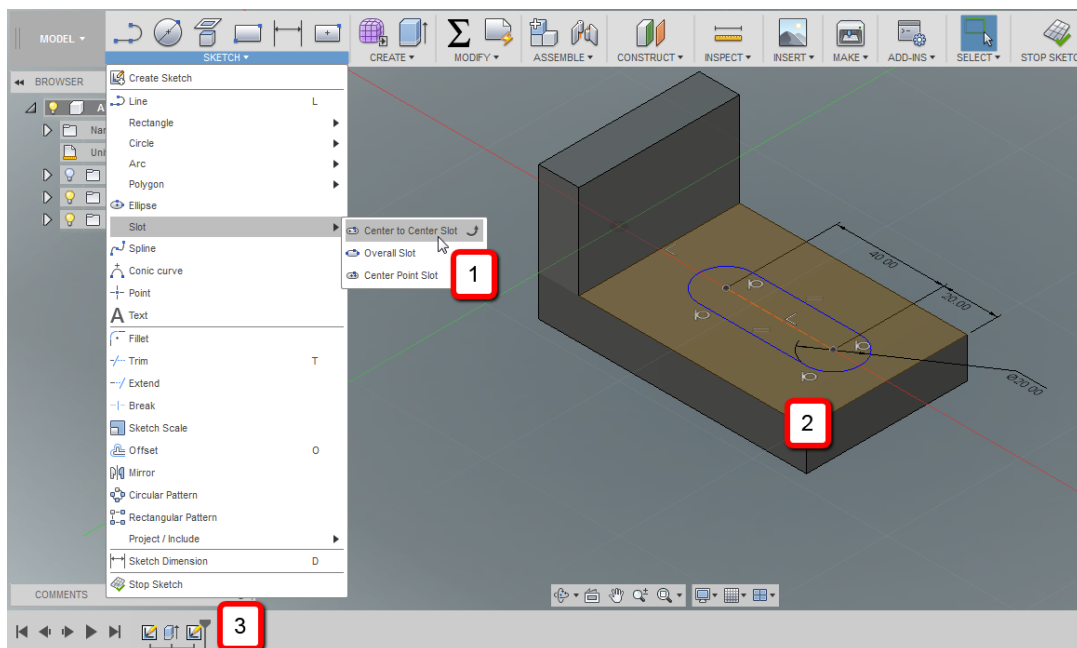
*ADDING A SKETCHED FEATURE TO THE FIRST BODY*

Adding placed features

The modify menu (1) is home to most of the "placed" features, or features that do not require a sketch, such as our example of Fillet (2) and Shell (3). These commands work directly on the model body in the context of the existing shape. In this example, fillets are placed by selecting edges and entering a radius, and the shell is created by selecting a face to remove along with a thickness. No sketch required.

TIP:
The fillet is placed first to allow the shell to use the rounded faces on the body.
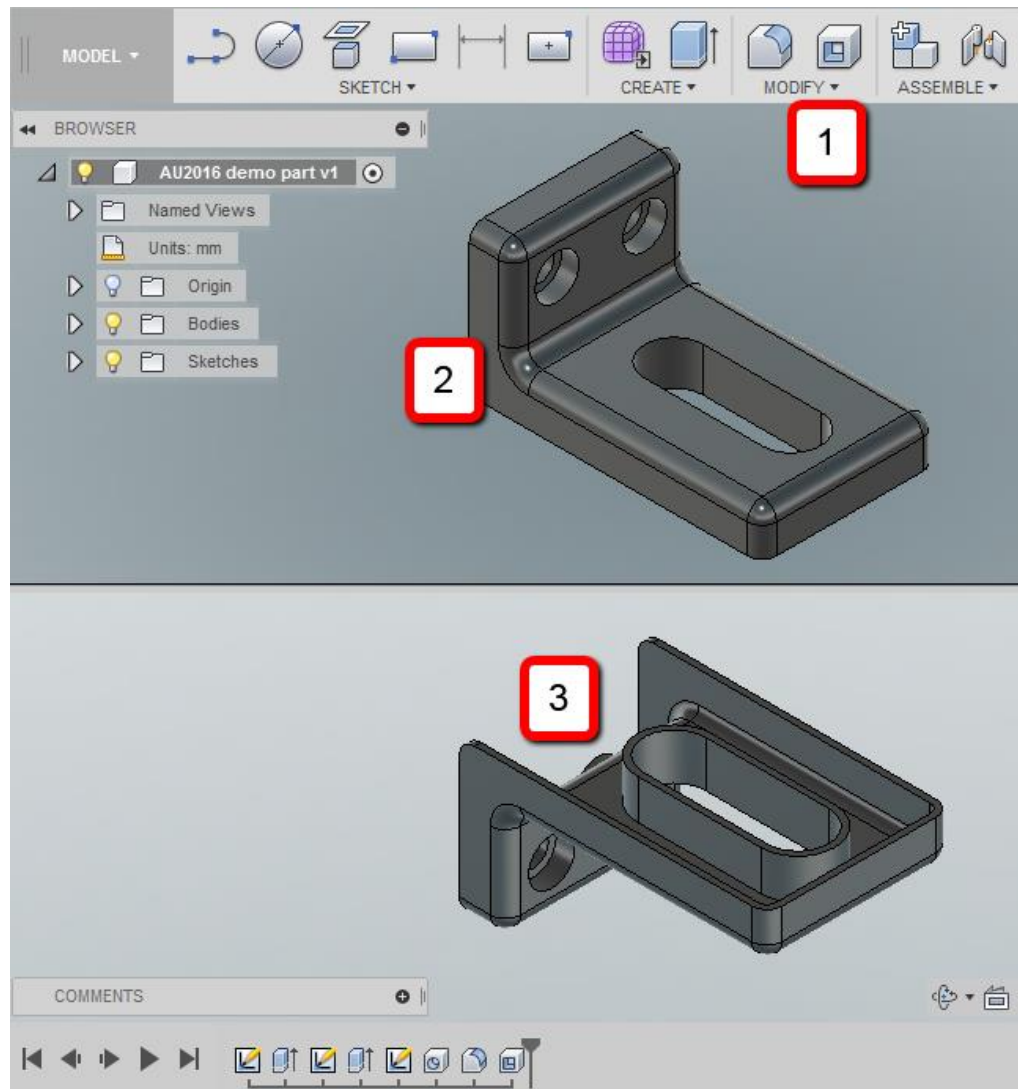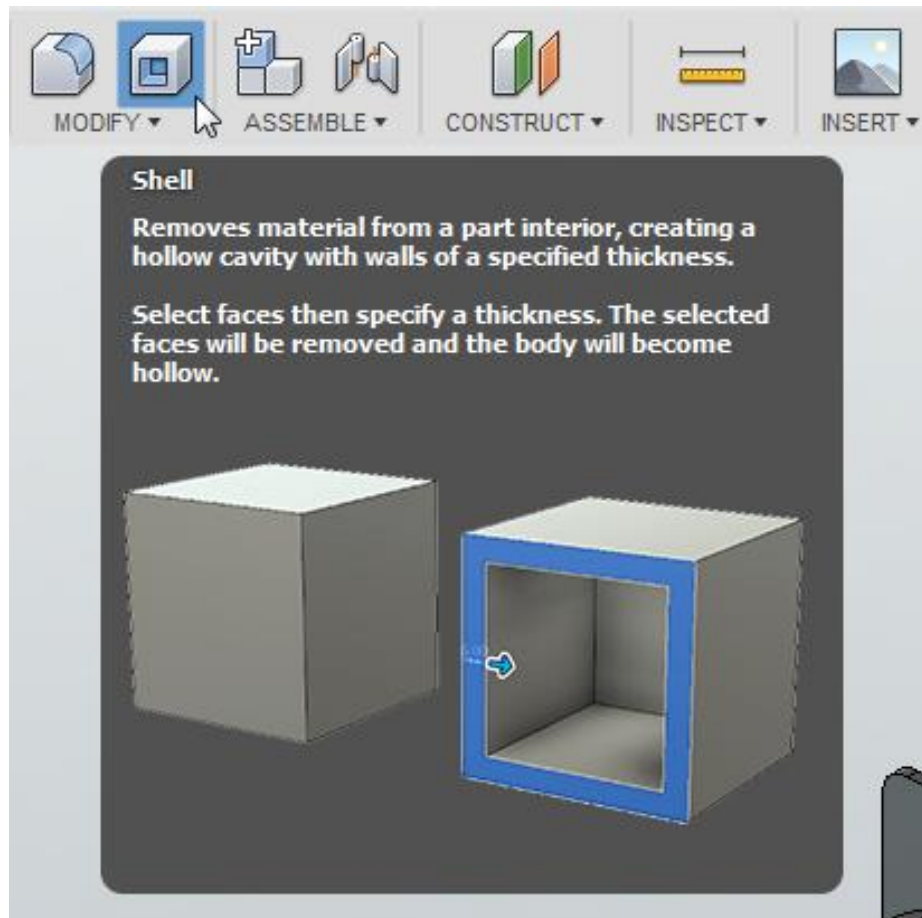

PLACED FEATURES: FILLET AND SHELL WORKING TOGETHER
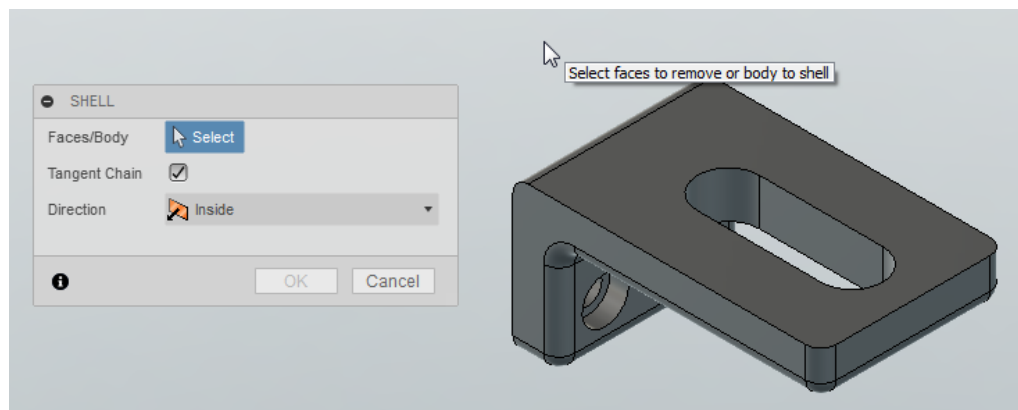
Tool tips

If you want to know what any Fusion command does, just hover over it and wait a second for the expanded tool tip.

TOOL TIPS

Command prompts

If you need a hint while using a command, stop moving your mouse and a command prompt will appear.


COMMAND PROMPTS

To conclude this portion I want to go a little deeper and talk a little about parametric vs non parametric modeling in Fusion 360 (which defaults to a parametric mode when you install it). As you would expect, like other parametric CAD programs, you can add your own parameters and formulas to the design. Or, as you model, just edit features and sketches to access your model's parameters.

TIP:
The Browser accumulates the things you make; the Timeline accumulates your actions. *Separating time and objects is unique to Fusion.*

There is another side of Fusion 360 however: history-free Direct Modeling. You can make Fusion start this way in preferences, or convert any parametric design by a right click command made on the top node in the browser: **Do not Capture Design History**, which will discard all of your parameters.

This mode, sometimes called DM (direct modeling) is great for simple parts and quick design iteration. You can always convert to a parametric document again, and you can mix the two kinds of parts in assemblies with no problem. Even in a parametric design you can use DM by invoking **Create Base Feature** to make a non-parametric modeling node on the fly. You simply do not always need parameters, so Fusion can accommodate however you embrace that.
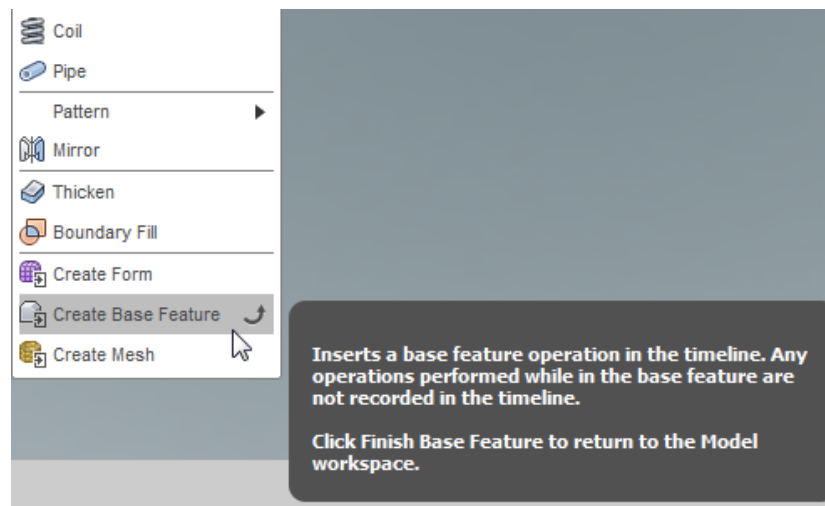
CREATE BASE FEATURE IS ON THE CREATE MENU

Related AU 2016 classes:
PD20709 - Fusion 360 Modeling Tips from the Experts

PD16277 - Use Direct Modeling in Fusion 360 to Take Your Models to the Next Level

PD20930 - Product Surfacing with T-Splines and Parametric Solid Modeling Tools

PD17918 - Creating Fusion 360 Custom Commands (API)

Creating assembly relationships in Fusion 360

When it comes to assemblies Fusion has several tools to help create relationships between components (parts). Each tool acts differently based on the task at hand.

- *Rigid Groups* are made for clumps of parts that always move together, like glue for sub-assemblies.

- *As-Built Joints* are used when parts are already in the right location and you must define their movement.

- *Joints* are used when you need to both position the components and provide control over how they move relative to each other.

Rigid Groups

Using imported assemblies as an example, you may just want to group the parts into a "block" that acts like one part, like for a Raspberry Pi board you inserted into your drone model.

The model imports as a group of components: PCB, connectors, chips, etc. However, Fusion considers every assembly as flexible. Meaning, you can move all the parts in the Raspberry Pi by dragging them. Since this is not a mechanism that has motion between its parts, Rigid Group should be used to glue the assembly into a single unit.
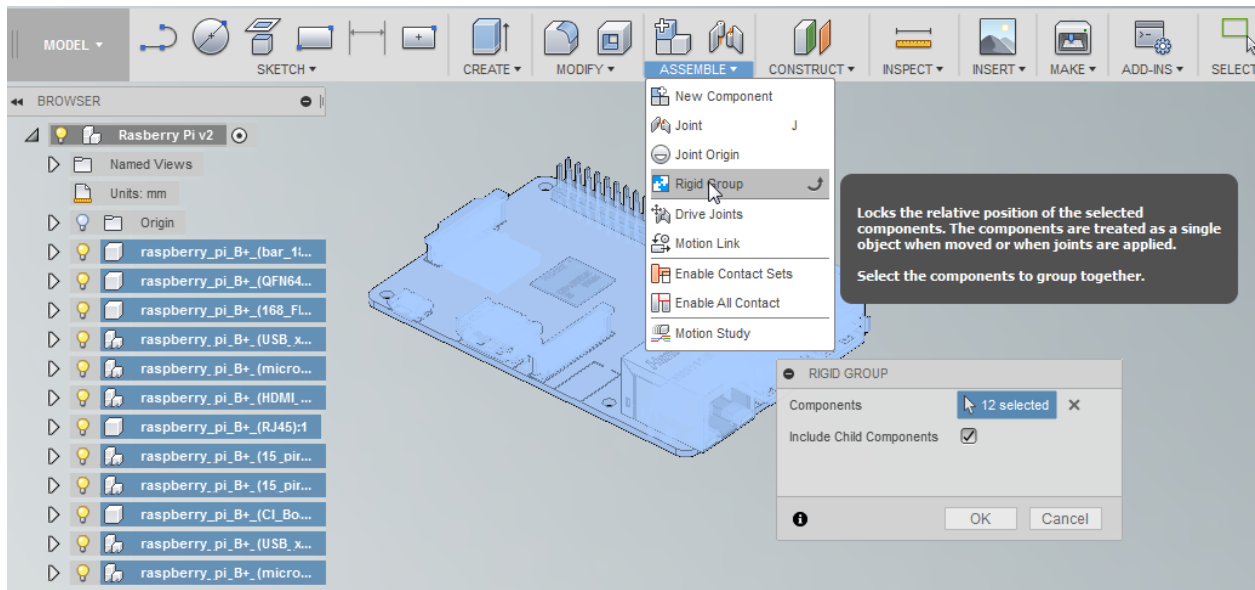
USING RIGID GROUP

Using As-Built Joints

When components are already positioned correctly relative to each other, and they require movement to be defined between them, a handy tool is the As-Built Joint. This command applies a regular joint between the components without forcing you to move them apart.

In the example shown below, clamshell cosmetics case is modeled as one body. The single body is then split into two and a hinge is designed while the parts are positioned together. An As-Built joint is applied to the parts in situ.
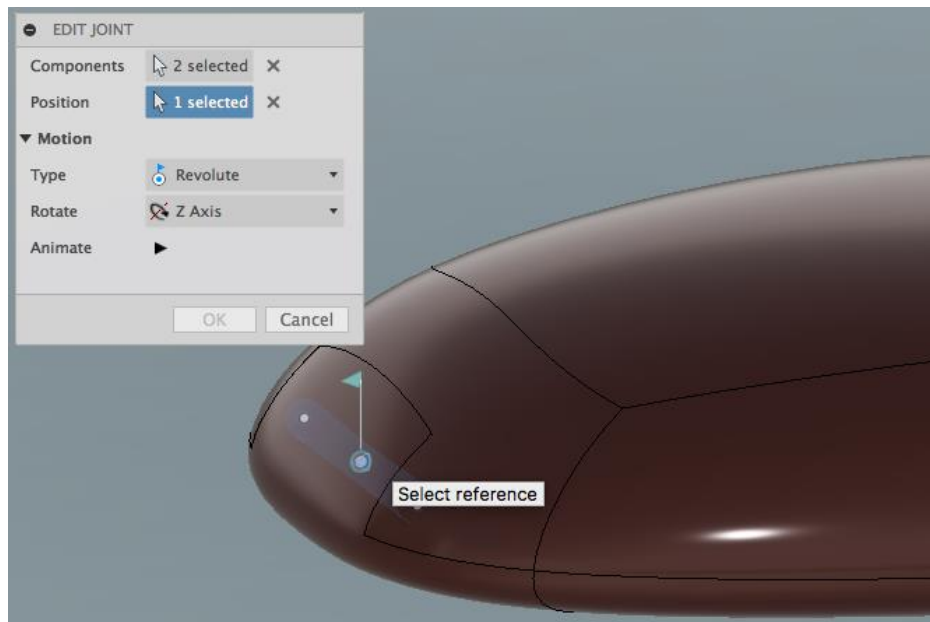
PLACING THE AS-BUILT JOINT WHILE THE PARTS ARE TOGETHER
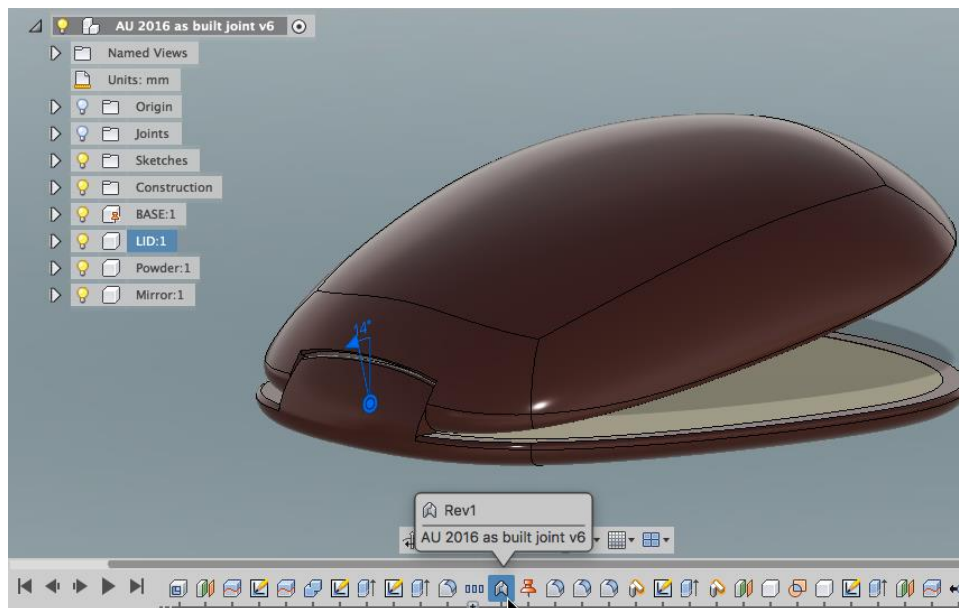
USING THE AS-BUILT JOINT

Using Joints to define position and motion together

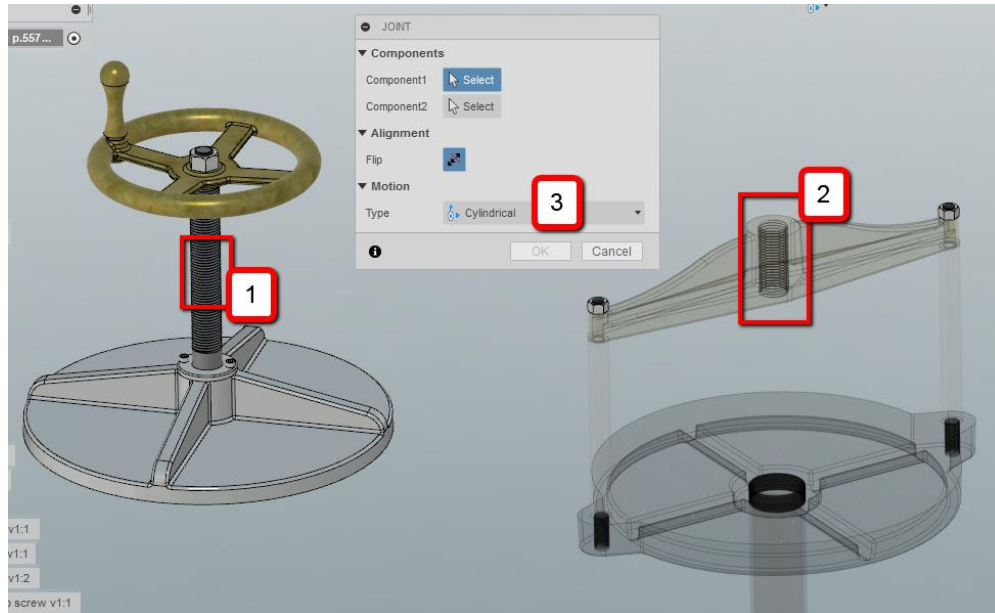The regular Joint command is used when parts must simultaneously move into position and have movement to define. This is the same set of joints used in As-Built, so the only difference is you're adding the step of moving them into position.


USING A JOINT TO POSITION A MOVING PART

In the image above, the sub-assemblies on the left and right already have Joints holding the parts in place. What must happen now is to define how the groups of parts interact, and also move the turning handle assembly to the press assembly. The threaded shaft (1) is to be joined to the press assembly (2) with a cylindrical joint (3). This type of joint allows for sliding motion along an axis, like a shock absorber.

TIP!

You can add information about a joint's range of motion, such as where it starts and stops, by using Edit Joint Limits, which is found next to the joint in the browser.
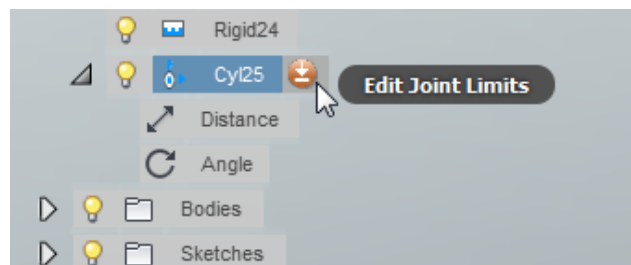
EDIT JOINT LIMITS

Joints are a deep subject with far too many details to cover in this Fusion 101 class. I'm trying here to show you the basics of the workflow so you know what to expect in your personal Fusion 360 journey of discovery. Don't forget to use the tutorials available on the learning page.
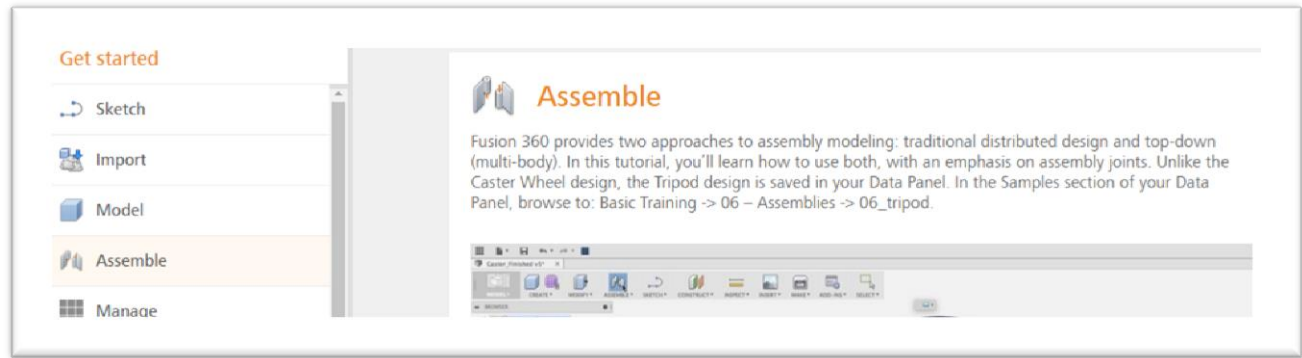
JOINTS ARE PART OF ASSEMBLE LEARNING

Related classes:

This is a very deep dive on Joints, from last year at AU 2015.
CP10165: Joints, So Much More than Just Rigid *Presented by Joel Palioca and Sachlene Singh*

A must see **AU 2016** class for anyone doing assemblies in Fusion 360:
PD19577: Fusion 360 Assemblies – Master Class

Make it real, or at least look really real

The last topic in this Fusion 101 class is about conceptual output from Fusion. How do you communicate your design? For some this is a nice set of realistic rendered images, for others a physical object is required. In this class we'll discuss the hobbyist path: renderings and printing.

If you need do create drafted 2D documentation for manufacturing your design Fusion has a drawing manager which is covered in depth here.

Render with cloud power or render locally
Fusion 360 has a powerful "local" render engine, meaning that your computer hardware performs all the calculations to produce a realistic photo image of your design. Fusion can also leverage cloud computing for rendering images, and thus leave your personal machine free for your other work. Anyone who has waited hours for a local rendering, you know the pain of having your machine tied up the whole time. Cloud rendering solves this problem.

The local render – quick and to the point
The advantage of local rendering, AKA in-canvas render, is a quick turnaround on a simple image. Most computers can produce good results in a couple minutes. Plus you have the advantage of just "point and shoot" style photography. Just rotate the model into a good view, change any settings that need changing, and let is bake. The image will continue to refine until you stop it.
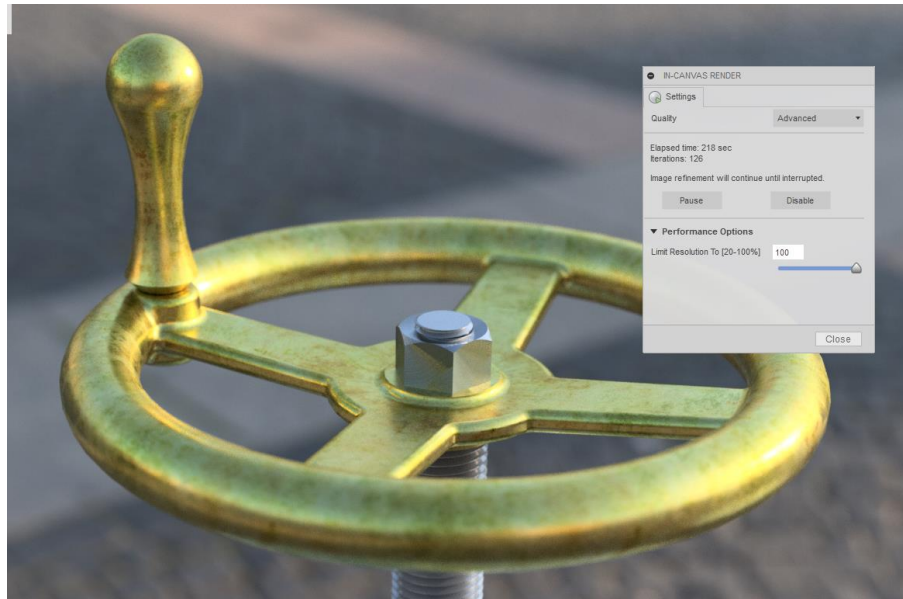
IN-CANVAS RENDERING

Render using the Cloud – let me use my computer!

You might decide to try a few local renders with the Quick setting to dial in your scene, shadows, depth of field and so forth. That takes a few seconds and doesn't tie up your computer for a long compute time. After you are confident your image is ready for prime time, fire up Cloud Rendering. An in-depth discussion of cloud rendering can be found in the Fusion community blogs.
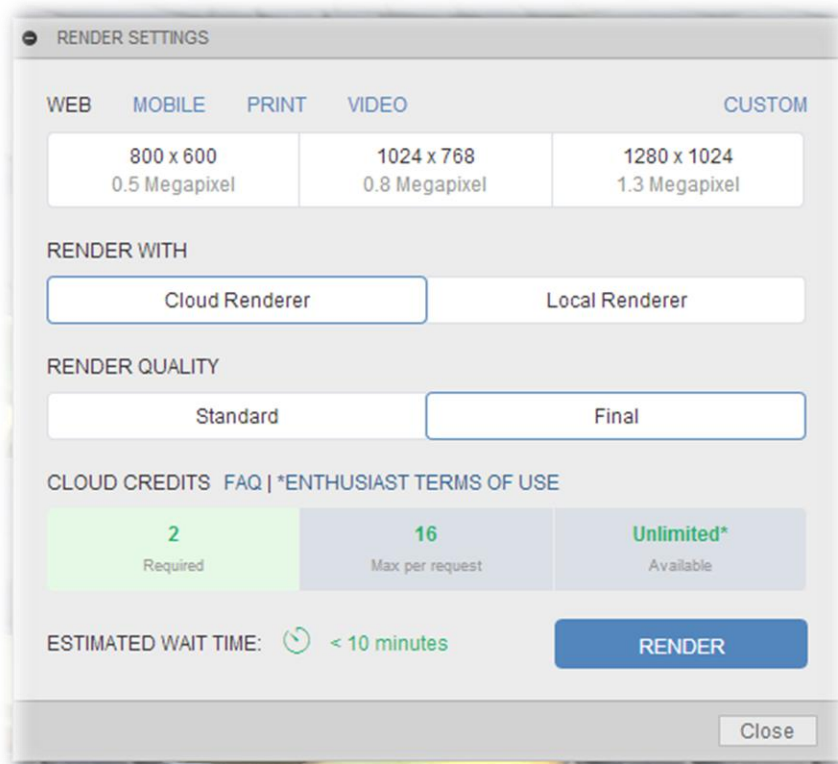
THE CLOUD RENDER DIALOG

Hold it in your hand today – 3D Print it!
If a photo realistic rendering is worth a thousand words, certainly a 3D print of your part is worth just as many. Sometimes feeling is believing, and your design needs to be *made* to be *real*. From inside the modeling environment Fusion 360 connects directly to most, if not all, commercial 3D printers. Go from designing to printing in a couple clicks.
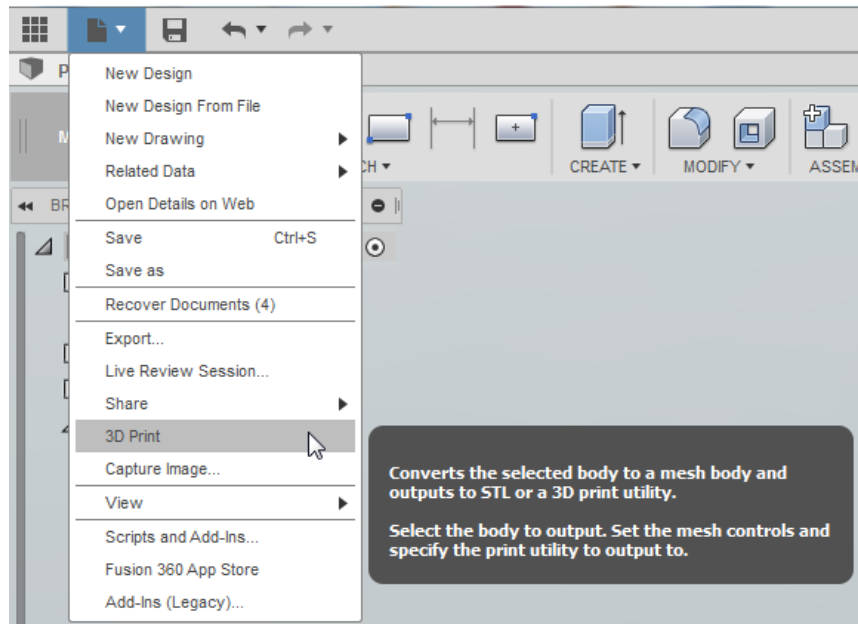
ACCESS 3D PRINTING FROM THE FILE MENU
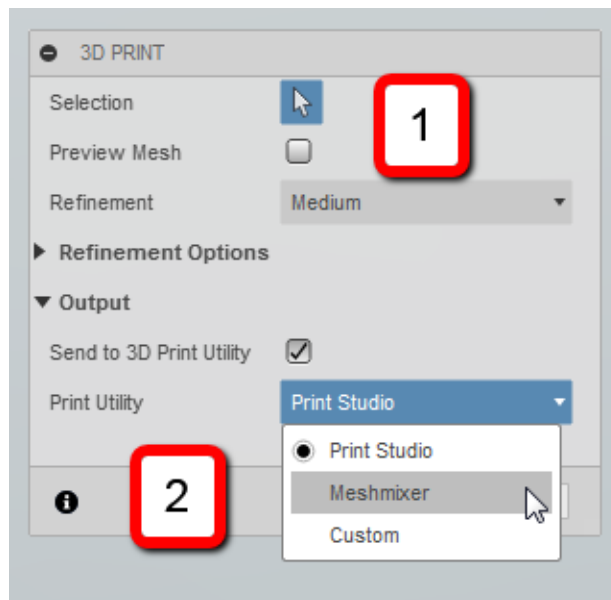
THE 3D PRINT DIALOG

Work with your favorite print utility
You may already know a lot about 3D printing, so if you have a preference for an app, do not worry, Fusion will recognize it and let you use it. Just start the 3D printing command, pick which body or component you wish to print (1) and which utility you wish to use (2).

Related AU 2016 classes(for rendering and 3D printing):
[PD17798 - It's all in the Details: T-Splines Modeling and Photorealistic Rendering in Fusion 360](#)
[MP20599 - 3D Printing and Prototype Development with Fusion 360](#)

Making it real extended, what else is possible? Find out at AU 2016!
We have only just begun to cover the options you have for making your designs real. From sketch to model, from part to assembly, from picture to print, these steps described in this class cover the most basic path for your design to make it into reality. As you work with Fusion 360, you will discover a world of tools and connected workflows, of data and communication, and a thriving community working alongside you.

Fusion 360 puts your data at the center. You need one model to simulate, to CNC, to make drawings of, and to share with your stakeholders. This is your Fusion design. It can do all of these things and more. Now that you have seen what Fusion is, go to these classes to find out what it can do for you!

CAM
[PD17417 - Fusion 360 CAM for CNC Programming Production Parts](#)
[PD21497 - Using Fusion 360 CAM Lathe Functionality in the Real World](#)

SIMULATION
[PD21984 - Simulation in Fusion 360 - Don't Get "Stressed" Out](#)

(Fusion 360 at AU 2016 cont'd.)

DRAWINGS
[PD21406 - "Here, Manufacture This!" Fusion 360 Drawings for Manufacturing](#)

WORKFLOWS
[PD21040 - Concept Design Workflow Guide for Industrial Designer with Fusion 360](#)
[PD19777 - Branch and Merge Workflows for Fusion 360 - Master Class](#)
[PD20476 - Transform your Approach to Product Development with Fusion 360](#)

PRODUCT DESIGN SUITE CUSTOMERS
[PD21025 - Maximizing Your Product Design Suite Subscription with Fusion 360](#)

Appendix A

To install Fusion 360 in a lab so all students can run Fusion

1. Start here: This [video on YouTube](#) that does a great job guiding you through the process of creating an account and downloading software on the new Autodesk Education Community.

2. Create an account (as an Educator or IT person) or login into Autodesk Education Community http://www.autodesk.com/education/ (this is the site where schools, students (13 and older), and teachers can access Autodesk software for their personal computer (for noncommercial use) or software for their school.

3. When creating your account, you can select if you are a student, educator, or IT.

   - Student: Can only get software for their personal computer
   - Educator: Can get software for their personal computer and / or software for their school
   - IT: Can only get software for their school

4. Once logged in, you can download Autodesk software from http://www.autodesk.com/education/free-software/all

5. For Fusion there are two installation options; installation for individual use, and installation in a computer lab (Fusion 360 for Educational Institutions). On the site you will receive instruction on the lab install.

   Fusion is licensed to the individual and not the school, but the Usage Agreement gives schools the right to install Fusion.

Free education license products*

A	B - H
3ds Max	Building Design Suite Ultimate
3ds Max Extension	CFD Design Study Environment
3ds Max Design	CFD Flex
Advance Concrete	CFD Motion
Advance Steel	Character Generator
Alias AutoStudio	Dynamo Studio
AutoCAD	Entertainment Creation Suite Ultimate
AutoCAD Architecture	Fabrication CADmep
AutoCAD Civil 3D	Fabrication CAMduct
AutoCAD Design Suite Ultimate	Fabrication ESTmep
AutoCAD Electrical	Factory Design Suite Ultimate
AutoCAD for Mac	Flow Design
AutoCAD LT	FormIt 360 Pro
AutoCAD LT for Mac	Fusion 360
AutoCAD Map 3D	Fusion 360 for Educational Institutions

FREE SOFTWARE FOR EDUCATION

 Note: Fusion 360 is updated approximately every two to six weeks and regular updates to lab installations are mandatory. You can sign up to be notified when an update is available.

Three easy steps to get up and running:

1. Download the Fusion Lab Install Package
   - Download for Windows
   - Download for Mac
2. Execute the Fusion Lab Install Package on each client
   - Run the executable from Windows Explorer, a command shell, script, or software distribution or management system.
   - The executable will prompt for administrator privileges if it does not already have them.
3. Sign-up to receive Fusion 360 Updates
   - Fusion 360 is updated approximately every six weeks. Regular updates to lab installations are mandatory. Maintain full compatibility by signing up for notifications.

   Click here to be notified

   You will receive an email with these instructions to upgrade Fusion 360:
   1. Re-download the Fusion Lab Install Package (link in Step 1 above)
   2. Windows only:
      - Download this batch file into the same folder/directory as the package.
      - Launch the *.bat file from Windows Explorer, a command shell, script, or software distribution or management system.

SIGN UP FOR UPDATE NOTIFICATIONS

6. Here is the direct link to install Fusion in a lab environment:
   http://www.autodesk.com/education/free-software/fusion-360-for-educational-institutions

Creating Autodesk IDs and logging in for students

1. Each student (13 and older) and educator will need to create an Autodesk ID on the Autodesk Education Community: www.autodesk.com/education

2. Use this Autodesk ID to log into Fusion 360

3. Each student will need to activate Fusion as an Education License

   http://knowledge.autodesk.com/support/fusion-360/troubleshooting/caas/sfdcarticles/sfdcarticles/How-to-activate-start-up-or-educational-licensing-for-Fusion-360.html

4. To help you and your students learn the Autodesk software, you can access Autodesk Learning Resources from www.autodesk.com/designacademy