Fusion 360

Mastery Bible

From Beginner to Pro

The Complete Step-by-Step Guide to CAD Modeling, Simulation, CAM, Assemblies, Drawings, 3D Printing, Electronics & Manufacturing

Gally Renders

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INTRODUCTION

Every product begins as an idea.

A sketch on paper. A rough concept in the mind. A problem waiting for a practical solution. But in today's world of engineering, product development, and digital manufacturing, ideas alone are no longer enough. To compete, create, prototype, and produce effectively, you need the ability to transform concepts into precise digital models, test them intelligently, refine them efficiently, and prepare them for real-world manufacturing.

That is exactly where **Fusion 360** becomes a game changer.

Fusion 360 is far more than a standard CAD program. It is a powerful, integrated platform that brings together **3D modeling, sketching, assemblies, simulation, rendering, drawings, CAM, CNC preparation, additive manufacturing, electronics design, collaboration, and workflow customization** in one unified environment. For students, engineers, designers, makers, manufacturers, entrepreneurs, and innovators, it provides the tools needed to move from concept to production with remarkable speed and precision.

Yet for many new users, the first encounter with Fusion 360 can feel overwhelming.

The interface is rich with commands. The workspaces are extensive. The possibilities are enormous. Sketches, constraints, parameters, components, joints, simulations, toolpaths, renderings, electronics, APIs—without a structured roadmap, it is easy to feel lost before you ever begin. Many users do not struggle because they lack talent. They struggle because they lack a guide that explains the software clearly, logically, and in the right sequence.

This book was written to be that guide.

Fusion 360 Mastery Bible: From Beginner to Pro is designed to take you on a complete learning journey—from the essential foundations of the interface and sketching environment to advanced workflows in modeling, assemblies, simulation, CAM, rendering, additive manufacturing, electronics, and customization. Rather than giving you fragmented lessons or disconnected tutorials, this book provides a structured path that helps you build skill upon skill with confidence.

You will begin by learning the fundamentals: how Fusion 360 works, how to navigate its workspaces, how to create and constrain sketches, and how to build accurate 3D forms. From there, you will move into more advanced topics such as construction geometry, feature-based modeling, patterns, fillets, lofts, sweeps, parametric control, assemblies, animation, and technical drawings. As your confidence grows, you will explore manufacturing workflows, CAM and CNC strategies, simulation and FEA, rendering and visualization, sheet metal design, additive manufacturing, PCB and electronics integration, collaborative workflows, and even the use of scripts, add-ins, and APIs to extend the software.

In other words, this is not a surface-level handbook.

It is a serious, in-depth resource built to reflect the true scope of Fusion 360 and the real demands of modern design and manufacturing. Whether your goal is to design consumer products, prototype mechanical systems, prepare CNC toolpaths, create production drawings, develop printable models, integrate electronics, or simply become highly capable in one of the world's most versatile design platforms, this book is meant to equip you with the knowledge to do so.

Just as importantly, this book is practical.

Throughout these chapters, you will not only learn what tools exist, but also why they matter, when to use them, and how they connect within a professional workflow. You will see how sketches drive models, how parameters drive design intent, how assemblies drive motion, how simulations drive improvement, and how manufacturing tools drive production. By the time you reach the later chapters, you will not simply be using Fusion 360—you will be thinking like a designer, engineer, and maker who understands the full design-to-manufacturing pipeline.

This book is for:

- beginners who want a clear starting point
- students who need a strong technical foundation
- makers who want to prototype and build
- designers who want to model with precision
- engineers who want to simulate, document, and manufacture
- professionals who want a broader, more integrated command of Fusion 360

If you are willing to learn step by step, practice consistently, and stay curious, the reward is enormous. Fusion 360 can become more than just software in your toolkit. It can become the bridge between imagination and execution—the place where ideas take shape, problems find solutions, and innovation becomes tangible.

So whether you are opening Fusion 360 for the first time or returning to master it at a higher level, you are in the right place.

Open the software.
Study the process.
Build with precision.
Create with confidence.

Your journey from beginner to professional-level Fusion 360 mastery begins here.

PART 1

INTRODUCING FUSION 360 AND DRAWING SKETCHES

CHAPTER 1
INTRODUCING FUSION 360

Fusion 360, because of the one-of-a-kind capabilities it offers, is quickly becoming one of the most popular Computer Aided Design (CAD) products available on the market for casual users (hobbyists). It is a single, collaborative platform that is cloud-based and can be used for engineering, CAM, and design. Its capabilities are comparable to that of Inventor, SolidWorks, or Catia. The user can carry out the whole of the product design and development process on a single (cloud-based) software thanks to this functionality.

Autodesk, which is one of the largest businesses in the computer-assisted design sector, is responsible for the development of the CAD application known as Fusion 360. They are also responsible for the development of the well-recognized software programs AutoCAD, Inventor, and Revit, which are used by millions of working professionals worldwide. They are venturing into uncharted territory with the release of their most recent product, Fusion 360, which is a tool designed for hobbyists and other infrequent users.

What Is the Function of Fusion 360?

Cloud-Based

You should be aware that Fusion 360 is a cloud-based modeling program at this point. When a model that is now being worked on is saved, the file is not stored locally on the device that it is being worked on, but rather it is saved on a server. If the project has to be accessed offline, it may be exported onto a hard drive, which can also be used as a way for turning the model files into the OBJ and STL file formats. Since it is cloud-based, this also implies that one's models and projects can be seen online from any computer or device. This is a significant advantage. Users now have access to a more portable and remote method of assessing models while they are on the move, thanks to mobile applications that are accessible for both Android and iOS. This, in turn, fosters improved team cooperation. This eliminates a significant portion of the headache that is often associated with team collaboration on larger projects in which design choices, markups, and comments are continuously being exchanged. It does not make a difference what device you are using to view Fusion 360, what version of the application you have installed, or whatever file format you wish to work with. Fusion 360 stores all of this information inside itself, and a click of the "**Share Public Link**" button makes it simple to send files to other people. In 3D CAD software, the concept of exchanging data so easily is unheard of.

Manufacture

As a result of Fusion 360's inherent support for computer-aided manufacturing (CAM), your CAM design may be manufactured in real life on a machine that uses computer numerical control, or CNC for short. It provides tools to create cuts, either by utilizing the software's default tools or by allowing the user to manually input the qualities and specifications of a tool that the user has previously earned. In addition to this, it utilizes a method known as Adaptive Clearing, which enables significant chunks of the material to be removed.

After a toolpath has been constructed, you will be able to see a simulation of the process of the model being created and examine the slicing operation. Because of this, difficulties in the cut may be avoided, and any last-minute design adjustments can be changed before any real machining is carried out. Because it minimizes mistakes in design and decreases the amount of wear and tear on machines and equipment, efficiency may result in significant cost and time savings.

Part Modeling

Fusion 360 integrates some distinct working environments, each of which offers a unique set of customization possibilities and other capabilities. When you launch the software for the first time, a blank modeling plane is displayed to you since the Design environment is the choice that is set as the default. Within this environment, 3D modeling is then done, and one of the most impressive aspects of Fusion 360 is that it provides the ease of having direct, parametric, freeform, and surface modeling all in the same area. While modeling in one environment, switching from, for example, a freeform workspace to a parametric one requires nothing more than the click of a button. This is one of the features of Fusion 360 that will save you the most time.

Rendering

When a design is developed on any computer program, there is always a need to know what it would look like in the real world as an actual product. This is something that can be accomplished by using a rendering application. The chosen model can be seen in an extremely lifelike fashion thanks to the Rendering tool that is included in Fusion 360. This feature generates a real-world setting that is based on your precise preferences and then displays it. The user can customize the surface by adding material properties, such as wood, marble, metal, glass, and many more. It is possible to download additional plug-ins, such as Key Shot, that will provide an even more realistic final render.

Simulation

When digitally evaluating a design, simulation may be a vital tool for analyzing how the design responds depending on the forces and external pressures that are given to it. A model's performance in real-world scenarios, as well as whether or not it would break under certain loads or pressures, may be evaluated using simulation, which can also reveal whether or not the model is imbalanced.

Within Fusion 360, the following is a list of some of the variables that may have their values altered:

- Modal frequencies
- Static Tension
- Temperatures
- Buckling of the structure
- Stress nonlinear under static tension
- The effects of heat stress

This might be challenging for users without prior knowledge, but if it is used appropriately, it can be a helpful tool in the design process.

Installing Fusion 360

System requirements

System requirements for Autodesk Fusion 360	
Operating System	Apple® macOS • macOS 12 Monterey • macOS 11 Big Sur • macOS 10.15.7, or newer, Catalina **Note**: macOS 10.14 Mojave support was deprecated on March, 2022. (More details) Microsoft® Windows® • Windows 11 • Windows 10 (64-bit) • Windows 8.1 (64-bit) (until January 2023)**
CPU Type	x86-based 64-bit processor (for example, Intel Core i, AMD Ryzen series), 4 cores, 1.7 GHz or greater; 32-bit not supported Apple silicon processors require Rosetta 2 - see this post for more information.

Recommended specs for complex modeling and processing	
CPU Type	3 GHz or greater, 6 or more cores
Memory	8-GB RAM or greater
Graphics	Dedicated GPU with 4 GB or more VRAM, DirectX 11 (Direct3D 11 or greater)

Step 1: Sign up for a Fusion 360 License

The first thing you need to do is determine what kind of license you need for Fusion 360 and sign up for that.

- Fusion 360 Commercial Subscription
    - Before committing to a paid membership, users may take advantage of a fully functional trial period that lasts for thirty days.
- Fusion 360 for Educational Use (free for qualifying students and educators)
- Fusion 360 for personal use, hobby use (free, for non-commercial use)

- Fusion 360 for startup use (free for eligible venture-backed, angel-backed, or bootstrap startups that are less than 3 years old and have 10 or fewer employees.
    - **Take note**: You will be required to apply for a start-up license. It is not granted permanently, and the request may be turned down at any time.

Before a certain license may be activated, the specific sign-up procedure that corresponds to that license type must first be completed in its entirety. Visit your Autodesk Account after you have your Fusion 360 license to access all of the licenses that are currently available to you. If you do not already have an Autodesk Account, you will need to establish one first. Commercial subscribers who have acquired a license may also examine the subscription and invoicing information, as well as control the users of the license they have purchased.

Step 2: Download and Install Fusion 360

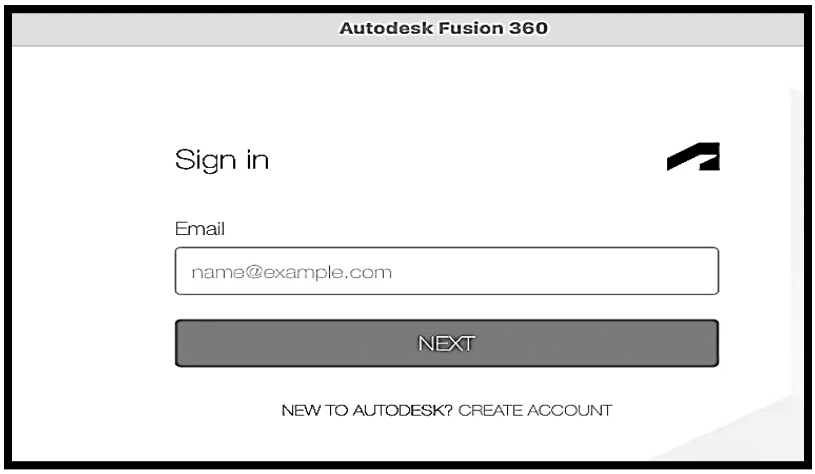
You are now able to download and install Fusion 360 once you have successfully secured your license for the software. Because it is a cloud product, Fusion 360 automatically downloads the most recent updates to operate on the most recent version.

To get started right now, just follow the steps that are outlined below:

1. Go to **manage.autodesk.com** and choose the option labeled **All Products and Services**
2. Select the product you want to download and then click the **Download Now** option. Since Fusion 360 is bundled with several different services, ensure that the **Download Now** button is selected.

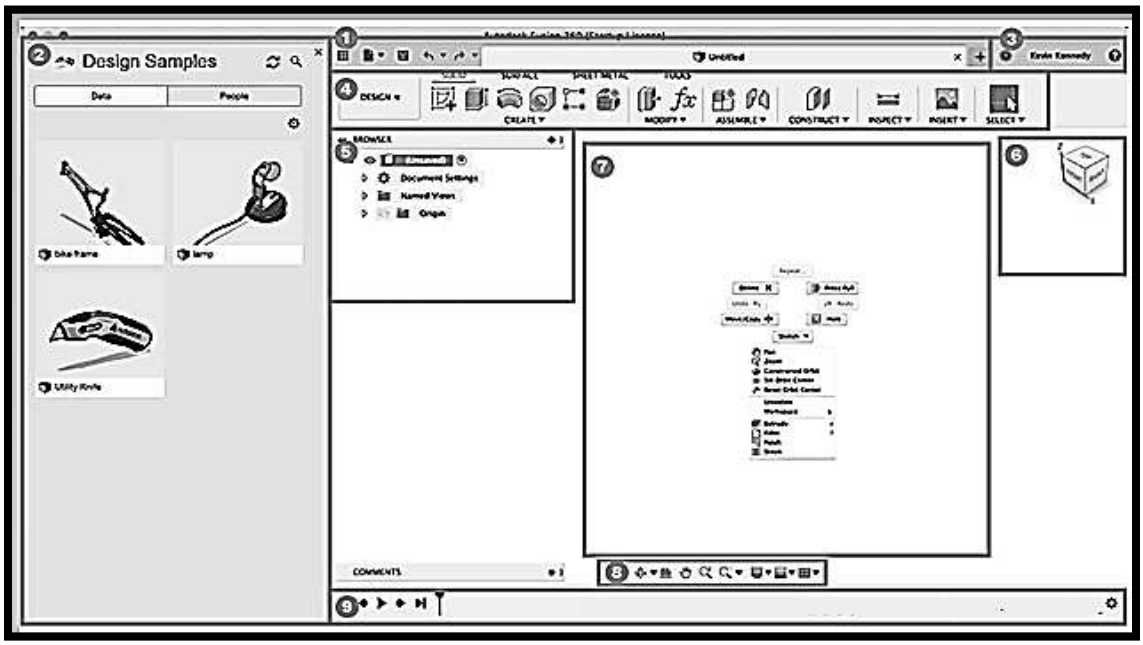
3. When you click the **Download Now** button, you will be sent to a new website where the installer will be downloaded without any more action required on your part. Note that the website can determine what operating system you are running and will either supply a Windows install file or a Mac install file depending on which one it determines you need.

4. When the file has finished downloading, you may begin the procedure by running the installer.
5. When the installation of Fusion 360 is finished, the program will launch on its own for the first time, and you will be prompted to sign in to your Autodesk account using the email address and password that you have associated with that account.



Getting Started with Fusion 360

Working with the User Interface of Fusion 360



1. Application Bar

The location of the Application Bar can be seen in the top left corner of the screen.

There are four primary components included inside the application bar.

- **Data panel** - contains your design files.
- **File menu** - create, export, or share your creations.
- **Save**: This allows you to save your designs together with descriptions of their versions.
- **Undo and Redo Buttons**: These allow you to undo or redo your most recent activities.

You will see tabs that reflect each design file across the top of the window. On the tab, you will see both the name of the file and the number of the version. On the other hand, if you have a big number of files open, you won't be able to view the name of the file until you hover over it. This operates in a way that is similar to how the tabs in your web browser function.

2. Data Panel

Your whole collection of design files may be found under the Data Panel. You may further organize your files by creating new projects and folders inside the data panel, which is located on the left side of the screen. You are also able to manage other users who are working on your projects using the data panel. However, depending on the kind of license you are using, there may be limits applicable to this functionality. You may open a project at any time by double-clicking on the file or right-clicking it and selecting the "open" option from the context menu that appears.

3. Profile and Help

Notification Center - Notifications will occur (a few times a year) with critical alerts, such as scheduled maintenance.

These notifications will be shown in the Notification Center.

- **Job Status** - View job status, Fusion 360 update status, and online/offline status.
- **Profile**: Click on your name to:
    - Access your Autodesk account.
    - Adjust your Fusion 360 preferences
    - Alternate between the two teams.
    - See or make changes to your profile.
    - Sign Out

4. Toolbar

The toolbar gives you the ability to choose the kind of working environment that best suits your needs. It is essential to keep in mind that the tools available on the toolbar will vary from one workspace to the next. There are additional tabs inside each toolbar, which further arrange the tools into logical categories. Tabs may be found in each toolbar. You will be able to personalize and rearrange the components of your toolbar as you get more familiar with your routine operations.

5. Browser

The Browser panel provides a summary of the document's settings as well as its views, as well as the origin, axes, and planes. To access the units, click the Document Settings button; here is where you may make adjustments to them. To show the model from the top, front, or side perspective, click the Named Views button. By clicking the origin, you can examine the multiple planes, which is helpful when picking an orientation for a drawing. There is one component that is always present in a Fusion document; this component is referred to as the **root component**. The top field of the browser displays the name of the browser, which reads Unsaved until you save it. You may access its functionalities by right-clicking on it. Everything that is added to the model, including drawings, bodies, components, and assemblies, is listed on the Browser panel as the model develops. You can adjust the display of these things and change them directly via the browser. Items are shown with names such as **Component 1:1, Component 1:2,** or **Body** as their default. The number that comes before the colon denotes the version, and the number that comes after it denotes the copy number. You can rename every entry in the browser, which is a useful practice for differentiating between them as their number increases. A new name can be entered by clicking the text box to activate it, typing it in, and then pressing the Enter key to complete the process. You may undock any panel by moving the bar at the top of that panel. To redock, move the panel to the border of the workspace, hold down the mouse button until a green line appears vertically, and then let go of the button. You may keep the browser window open, or you can minimize it by clicking the minus symbol **(-).**

6. Viewcube

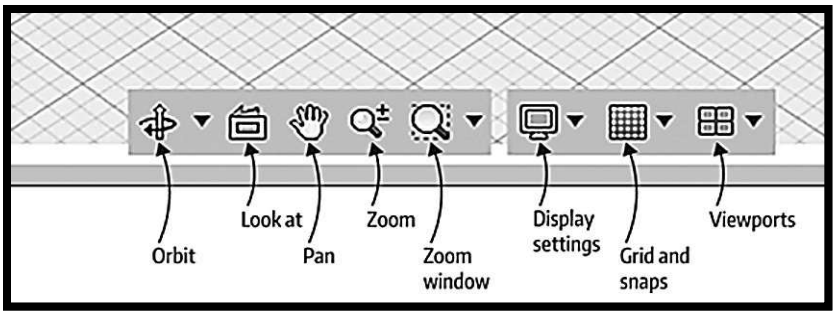
You can see your design from a variety of normal view locations as well as revolve around your design using the viewcube. You have the option of clicking and dragging the viewcube in any direction, or you may choose to choose certain faces, corners, or arrows. You may also see the model in the default home position by clicking the home icon, which is located next to the viewcube in the interface.

7. Canvas and Marking Menu

You'll be conducting all of your design work and sketching in the center portion of Fusion 360, which is where the app's name comes from. Because of this, we refer to this part of the picture as the canvas. The "**marking menu**," also known as the right-click menu, is accessible from inside the canvas and may be accessed by clicking the right mouse button. If you right-click, you'll be able to access commonly used commands, as well as the option to switch between different workspaces; this saves you from having to go to the top left corner of the screen. You'll need to commit the location of the various marking-menu elements to memory if you wish to make full use of the menu. Users have the option to right-click and drag (in the same direction) to fast reach the function that they wish to utilize. This is done via the marking menu. To activate the press, pull command, for instance, right-click the mouse and move it toward the two o'clock position.

8. Navigation Bar and Display Settings

You may zoom, pan, and orbit using the tools that are included inside the Navigation panel. In addition to that, it features display options that may alter the workspace as well as the way the model looks. You may, for example, alter the size of the grid, adjust the way the mouse snaps to the grid, see the model as a wireframe, and see many versions of it on the same screen. This section explains the various navigational tools. To see the available choices, choose the arrow with the downward pointing triangle next to each icon.



Orbit: This is a view that rotates the viewer around the model, allowing for a comprehensive examination of the object from a variety of perspectives. It is not the model itself that is moving, but rather your location about it. You have the option of orbiting freely or being bound. Orbits that are constrained around either the xy plane or the z-axis. Free to go in whatever direction the user chooses. To release orbit more quickly, you need first to depress and hold the Shift key, and then you should depress and hold the scroll wheel on your mouse.
Look At: First choose an area on the model to click, and then select this icon. The location of the model will adapt itself appropriately.
Pan: The model will move across the screen as you pan the camera. You can pan more effectively by depressing and rotating the scroll wheel of the mouse while holding down the mouse button.
Zoom: This provides you with either an enlarged view of the model (think of it as a telephoto lens) so that you can notice minute details or a reduced view of the model (think of it as a wide-angle lens) so that you can see the whole picture. To zoom in or out with more precision, just scroll the mouse wheel in the desired direction.
Zoom Window: This allows you to look at a certain spot in more detail by allowing you to drag a window around that location. When you choose the **Fit** option, the model will occupy the whole screen. If your model disappears after you click it, it's because there are still bits and parts of it that you sketched previously that are sticking around.

Find those files and delete them to make your model appear again. Using Clicking Fit, you may find "lost" components of your puzzle.

- **Display Settings**: This allows you to change the appearance of the working area.
- **Grid and Snaps**: You may choose whether the grid is shown or not, and you can also change its parameters and the snap increments.
- **Viewports**: Show the workspace either as one huge workspace or as many smaller workspaces at the same time using viewports.

9. Timeline

The timeline provides a listing of the operations in the sequence in which they were done on your design. The timeline is a row of icons that shows at the bottom of the screen if you are creating in a parametric manner. This row of icons is sometimes referred to as the history tree. There is a corresponding symbol for every activity that is carried out. The timeline expands along with the project; however, it may be made more comprehensible by grouping many icons (to do this, click the first icon in the timeline, then while

holding down the Shift key, and click the final symbol). If you double-click on an icon, the feature that corresponds to that icon will be selected in the model. If you right-click on an icon, you will have access to a context menu, and if you drag an icon to the left or right, you can change the order in which operations are calculated. By moving the slider to the left, you may "**travel back in time**," which means you can review actions that occurred previously. If a feature has a yellow indicator, it indicates that there is a caution associated with it; for example, you may have destroyed something that Fusion requires to maintain the integrity of the design, but Fusion cached it, which allowed you to continue working. A red indicator indicates that there is a problem; one example of this is that an edge that a fillet utilized has been erased, which implies that the fillet can no longer be created. It is in everyone's best interest to address warnings and mistakes as soon as they appear.

You can disable the timeline by right-clicking the title field of the browser and selecting the **Do Not Capture Design History** option from the context menu. This puts you in direct modeling mode, which, compared to parametric modeling mode, you may find to be more straightforward. To bring up the timeline again, right-click the title field of the browser and choose the **Capture Design History** option from the context menu. When you enter Direct Mode, the timeline will begin from that moment, and you will permanently delete any timeline icons that were there before you entered **Direct Mode**.

Invoking a New Design File

When you launch Autodesk Fusion 360, the program prompts you to create a new design file and gives it the default name "**Untitled**." Earlier on, many components of the starting user interface of the new design file were covered. These included the **Application Bar, Toolbar, BROWSER, and Timeline.** In addition to the design file that is loaded by default, you can load a new design file by selecting the New Design tool from the drop-down menu in the application bar that is labeled File. When this is done, a new design file with the name "**Untitled**" as the default name is called, and it is automatically set to the active state. You can also start a new design file by clicking on the plus symbol that is located next to the name of the current design file that is open. The tools that are accessible in the Toolbar change depending on the workspace that is now active. The **DESIGN** workspace is the one that is active by default. As a consequence of this, the tools that are required for the creation of 3D models, surface models, and sheet metal models may be found inside the different tabs of the Toolbar.

Working with Workspaces

DESIGN Workspace

You can generate and modify the solid, surface, and T-Spline model geometry in the Design workspace, which is controlled by the geometry of 2D sketches. This workspace is the one that most closely resembles a conventional 3D CAD environment. In this setting, you may develop history-based features (such as extrude, revolve, loft, and sweep) that adapt to changes in the design.

Sketch contextual tab

The Sketch tab is a contextual tab that provides tools that allow you to generate and change 2D drawings that drive the 3D geometry of a design. These sketches may be found under the tab named "**Sketch**." Within the Design workspace, the Sketch contextual tab may be accessed in a variety of different places. The most typical location to find it is under the Solid tab's Create panel under the Create Sketch menu item.

Since you can create a Sketch inside many tabs in addition to contextual tabs, Sketch is treated as a special case and shown with the other non-contextual tabs also visible. This is because you can build a Sketch within multiple tabs. When you choose Create Sketch, a new contextual tab titled Sketch appears. This tab has a dedicated toolbar that, by default, is stocked with the sketch tools that are used the most often. By default, the toolbar will additionally display the sketch limitations that are now active.

You will see that the tab itself as well as the button labeled Finish Sketch will be marked in blue to let you know that you are now operating in a temporary mode. In contrast to the other contextual tabs, the Sketch contextual tab does not prevent you from switching to other tabs while it is active. This is the primary distinction between the Sketch contextual tab and the other contextual tabs. This is because you can use modeling commands (like Extrude, for example) even while your sketch is currently active. Performing this action will cause you to be taken out of Sketch mode and into the command itself automatically.

Solid tab

Tools that enable the creation and modification of solid models may be found under the Solid tab.

Form contextual environment

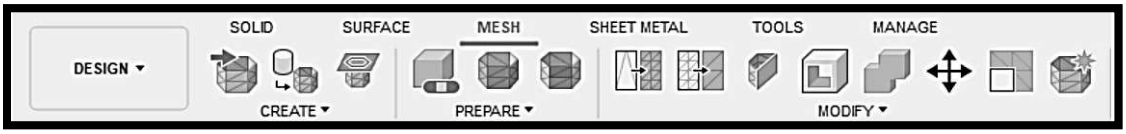
The form is a contextual environment that gives you the ability to push and pull T-Spline bodies' faces, edges, and vertices to form intricate organic features. The tools are analogous to those used for sculpting clay. The Form contextual environment can be accessed via the Create panel of the Solid tab in the Solid editor.

Surface tab

Tools that enable the creation and modification of complicated parametric surfaces may be found under the Surface tab. The exterior contour of a design is denoted by its surface, which is flat and devoid of depth. You may also use the tools on the Surface tab to patch or fix openings in a model. These tools are located on the Surface tab.

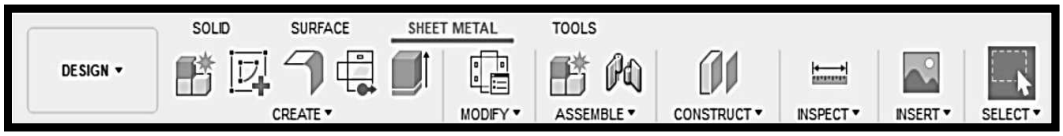
Mesh tab

Tools that enable the creation and modification of parametric mesh bodies may be found under the Mesh tab. A mesh body is a collection of polygon faces that are formed from vertices and edges of the polygons. A mesh does not have any thickness to it. In the process of additive manufacturing, meshes are often employed. You can insert new mesh bodies, repair existing ones, and change existing ones by making use of the Mesh commands to get them ready for manufacture.



Sheet Metal tab

You may construct and change components made of sheet metal by utilizing the tools in the Sheet Metal tab, which are organized according to sheet metal rules. Utilizing 2D drawings and various cutting processes, flat designs may be documented and manufactured using these methods.



Flat Pattern contextual surroundings

Flat Pattern is a contextual environment that enables the creation of a flat pattern for sheet metal starting from a folded design. Through the Create panel of the Sheet Metal tab, you will have access to the contextual environment of the Flat Pattern.

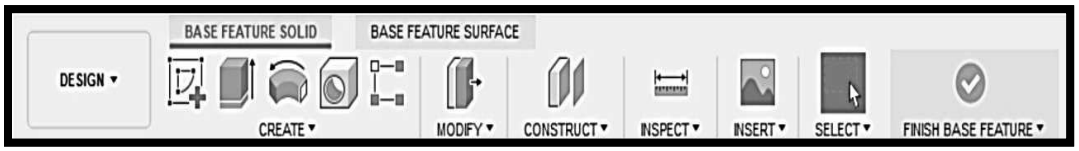
- **Flat Pattern Solid tab**: To build and change Flat Patterns with solid geometry, use the tools on the Flat Pattern Solid tab, which are located in the tab's name.

- **Flat Pattern Surface tab**: To build and change flat patterns with surface geometry, use the tools on the Flat Pattern Surface tab, which are located in the tab's name.

Base Feature contextual environment

Base Feature is a contextual environment that provides you with the ability to enter a direct modeling sandbox and inserts a feature in the Timeline that does not have any history associated with it. You get access to modeling tools as well as surfacing tools when you use the base feature. Through the Solid tab's create panel, you will have access to the contextual environment known as the Base Feature.

Base Feature Solid tab: To construct and change solid geometry, use the tools that are located on the Base Feature Solid tab.



Base Feature Surface tab: To develop and alter surface geometry, you may use the tools that are located on the Base Feature Surface tab.

Generative Design Workspace

You can develop various CAD-ready design solutions all at once with the help of the workspace known as Generative Design, which is determined by the manufacturing and performance criteria.

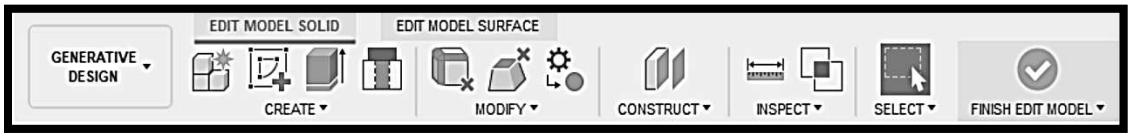
Define tab

You may build up design studies with well-specified objectives, limitations, materials, and production alternatives with the help of the tools that are included under the Define tab. Tokens may be used to generate several different design possibilities in the cloud that are process and performance conscious. Investigate and assess each design alternative based on the tradeoffs that are most important to meet your requirements. After that, export your ideal design, and then import the geometry that is suitable for CAD into Fusion 360.

Edit Model contextual environment

Within the context of the Change Model environment, you can utilize the usual modeling tools that come with Fusion 360 to build and edit model geometry, as well as to construct obstacles and preserve geometry. Through the Define tab's Edit Model panel, you will have access to the contextual environment of the Edit Model panel.

Edit Model Solid tab: The Edit Model Solid tab includes a set of tools that, when used, enable users to edit the geometry of solid models without having to leave the Generative Design workspace.



Edit Model Surface tab: This is where you'll find the tools that will allow you to edit the surface model geometry without having to leave the Generative Design workspace.

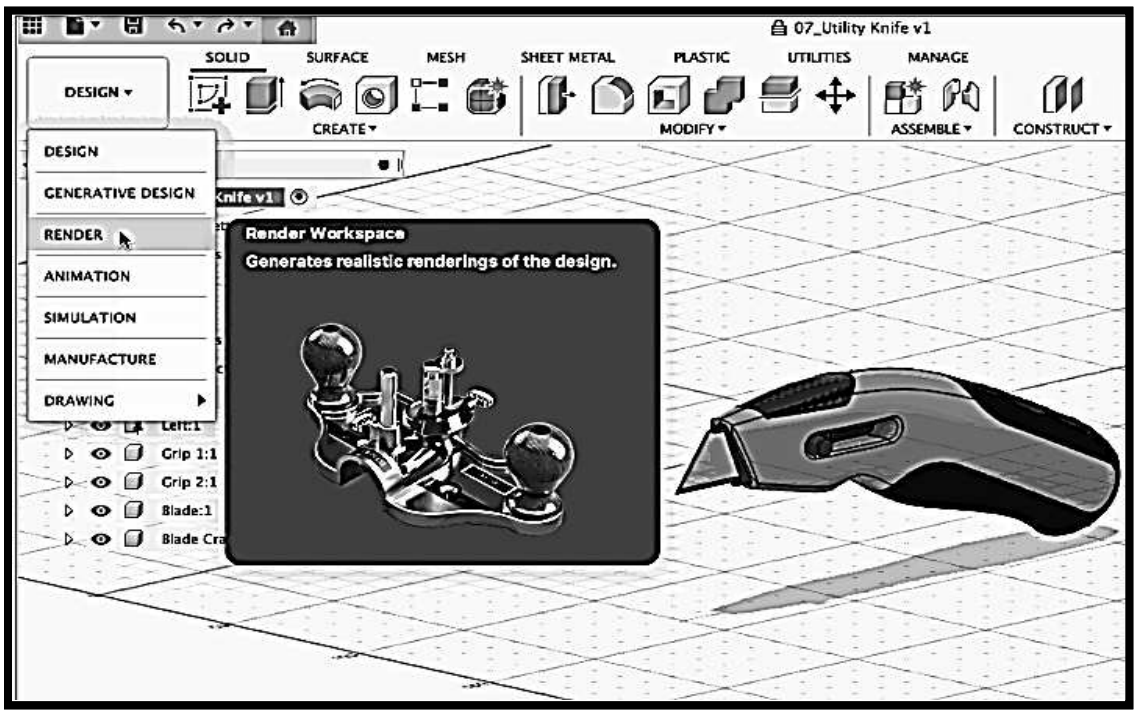
Explore contextual environment

You are given the ability to show the model, stress, export preview, and design space views of a result inside the Explore contextual environment. Additionally, you have the option to view the Comparison View. Through the Define tab's Explore panel, you will have access to the contextual environment known as **Explore**.

Outcome View contextual tab: Within the Explore contextual environment, the toolbar will switch to the Outcome View contextual tab when you select one or more outcomes to explore in the 3D View or Comparison View. This tab includes a variety of tools that allow you to display, compare, and export outcomes. When you select one or more outcomes to explore, the Explore contextual environment will begin.

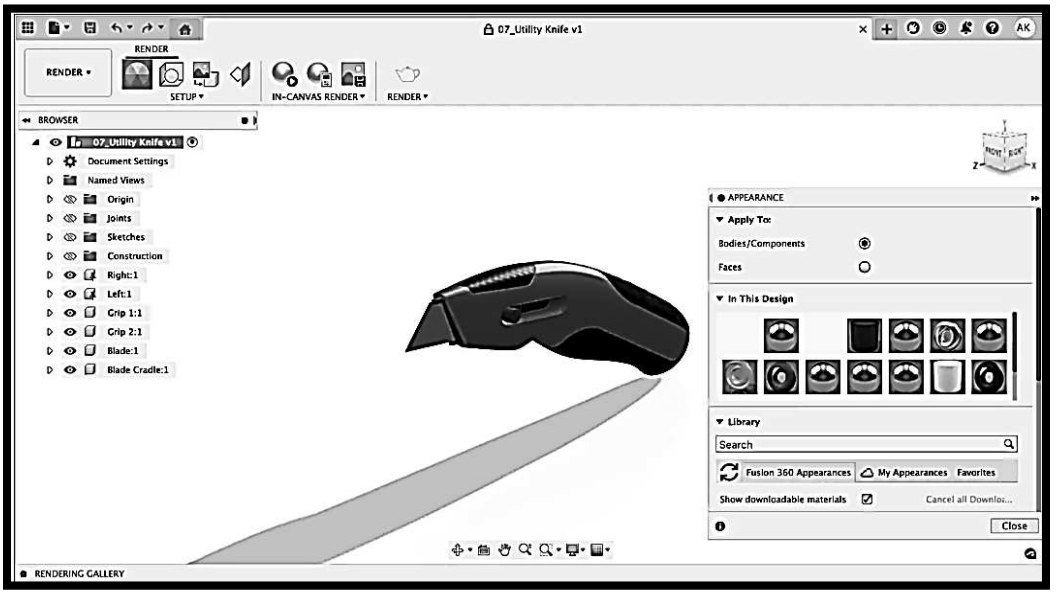
Render Workspace

You can get a more accurate and thorough picture of what your final product will look like by using renders. These renderings have the potential to serve as an invaluable resource for product marketing as well as the idea and depiction of products. To go over to the render workspace, just right-click on the current work environment you are in and pick **Render** from the menu that appears.

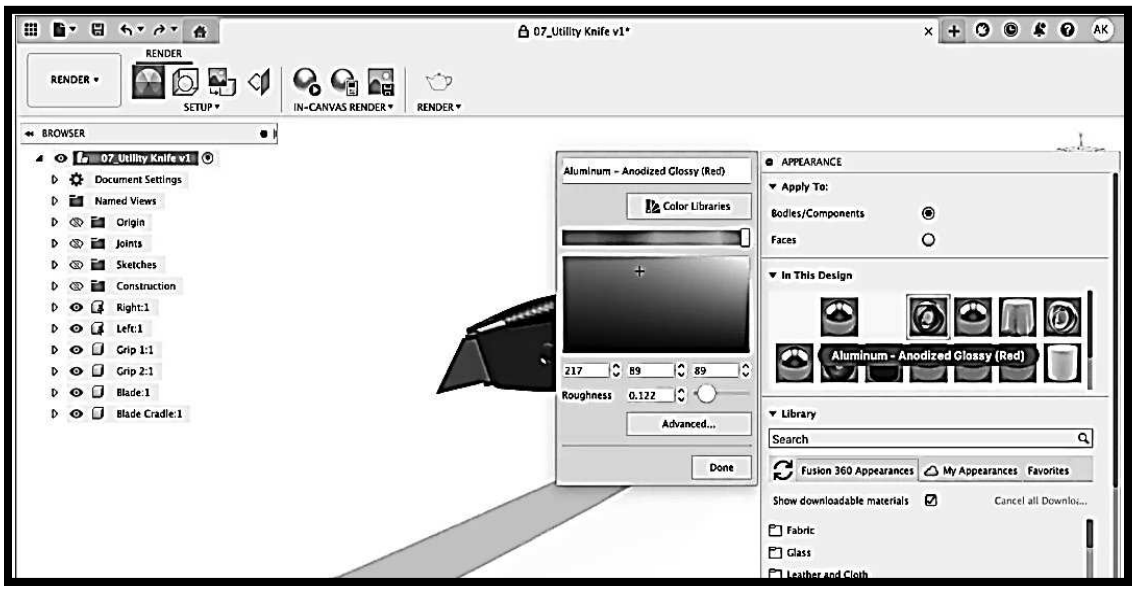


The appearance of your model will abruptly undergo a significant transformation if you apply this change. Let's look at the instructions to have a better understanding of how to make use of this area.

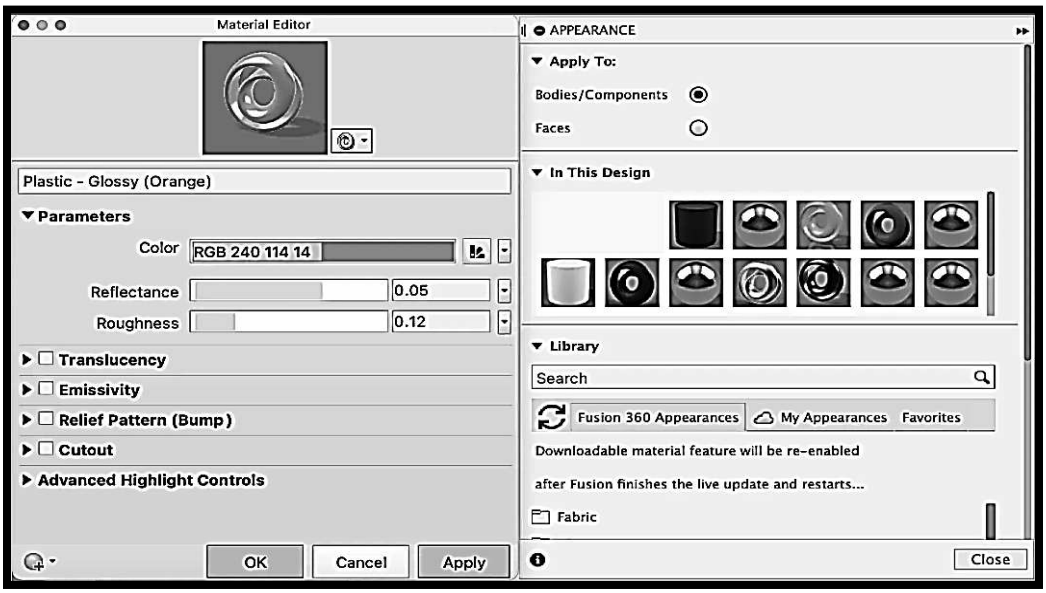
Appearance



You can change the kinds of materials that your model is made up of by using the Appearance command. Within this menu, in addition to the ability to choose a material from a list that includes leather, liquid, metal, plastic, wood, stone, and other alternatives, you can also modify the color, texture, and feel of your material.



You can edit the content by either right-clicking on it and selecting Edit from the context menu, or you can double-click on the part of the material that you want to change. When you click this button, you will be sent to a new dialogue where you will be able to modify the fundamental features of the look of your material. After you have finished the fundamentals, you may select the **Advanced** button to dive into more specifics about the texture, bump pattern, sheen, and feel of your material.



Understanding the Advanced settings dialog

- **Reflectance**: This is the amount of light that is reflected from a surface.
- **Roughness**: This refers to the degree to which the surface is abrasive, which in turn determines how glossy the surface looks.
- **Translucency**: This determines how much of the surrounding environment can be seen through the material.
- **Emissivity**: This is the property that transforms a substance into a source of light.
- **Relief Pattern (Bump):** This displays the bump map that has been applied.
- **Advanced Highlight Controls**:
    - **Color**: This allows you to alter the hue of the highlight. For the most part, you will want to keep this white for a more authentic appearance.
    - **Shaping**: Alternate between highlights that are smooth (Long Falloff) and highlights that are sharper (Short Falloff)

ANIMATION Workspace

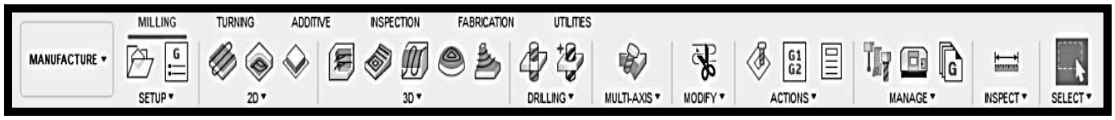
Your ability to describe your design using 3D exploded views and animations that illustrate design assembly are both facilitated by the Animation workspace. You may assist your teammates and customers in better understanding and assessing your design by sharing videos with them.

SIMULATION Workspace

You can put your design through its paces via the use of finite element analysis by setting up studies in the Simulation workspace (FEA). Simulate to see how your design fares under a range of different loads and situations. Analyze the data to better comprehend the physical constraints imposed by your design. Investigate a variety of design options, and then settle on alterations to the design based on your findings.

MANUFACTURE Workspace

The workspace known as "Manufacture" gives you the ability to design toolpaths for the production of your components using methods such as machining and turning (also known as "subtractive manufacturing") or 3D printing (additive manufacturing).



DRAWING Workspace

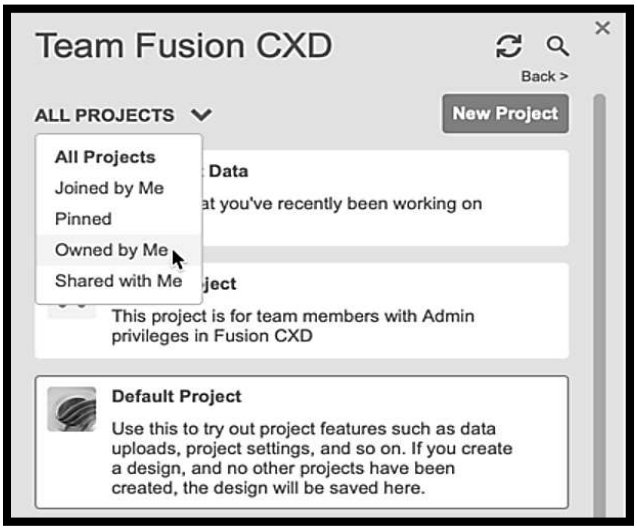
Documenting production requirements via the use of integrated, associative drawings and animations for both parts and assemblies is possible when you use the Drawing workspace.

Managing Data by Using the Data Panel

- You can get access to your Fusion 360 projects and design data, as well as organize and share it, by using the Data Panel.

- The Data Panel provides access to your designs and allows you to manage projects.
- By default, the Data Panel will not be shown. To access it, you need to choose the Show Data Panel icon from the toolbar.

A helpful hint is that you can reveal or conceal the Data Panel by pressing **Ctrl+Alt+P** (on Windows) or **Option+Command+P** (on macOS).



You will be able to examine both your projects and designs using the Data Panel. Additionally, it provides you with sets of samples that you may use, such as CAM samples.

Make the projects you want to see more or less apparent by using the project filter:

- **All projects:** the full list of projects in the current Team
- **Joined by Me**: only the projects you have joined
- **Pinned**: only favorite projects
- **Owned by Me**: only projects you started
- **Shared with Me**: only projects to which you've been invited

Join a project

It's possible to start or end a project. An open project welcomes participation from any member of the team. In a closed project, team members may only join if they have been invited by a project administrator or have been authorized by a project administrator. Even while everyone on the team can see open and closed projects on the list, to view the contents of a project, you will need to join it first, even if the project is open. To participate in an open project, position your mouse so that it is in the top right corner of the project, and then click the Join button. To participate in a project that is currently full, move your mouse to the top right corner of the project, and then click the **Request Access** button. Your request will be looked at by the administrator of the team. The project will display Access Requested until the administrator decides whether or not to grant you access.

Grant access to a project (Administrators Only)

When someone requests access to a project, you, as the administrator of the team, are told about it.

- Position your cursor so that it is in the top right-hand corner of the project, and then choose the **View Access Requests** option.
- Take a look at the permission requests.
- Either click the **Approve** button to allow the requester access or the Reject button to refuse the request.

Open a project

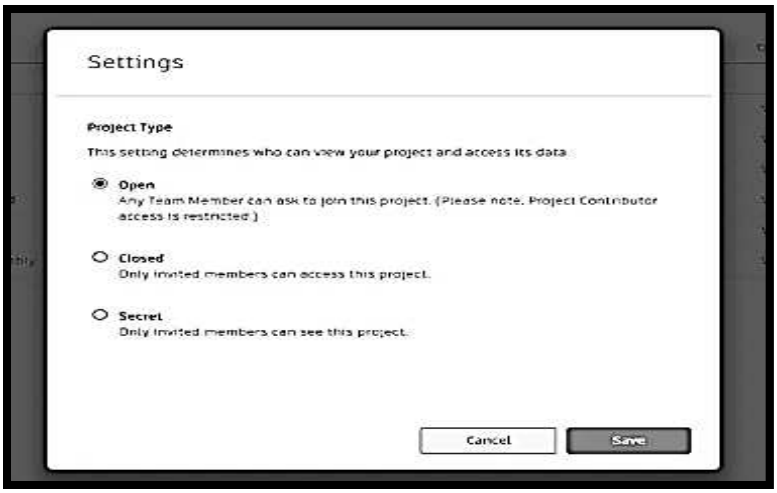
If you want to access the data that is stored in a specific project or sample folder, you can do so by double-clicking on the name of the project or sample. Take note that the name of the team is shown at the very top of the data panel. Simply go to your team inside the online application by clicking on the team's name.

Creating a New Project Folder and Sub-Folders

1. Within Fusion 360, first expand the Data Panel and then click the **Home** icon to navigate to the top of the data structure.
2. Select **New Project** from the drop-down menu, and a fresh entry will be added to the project list.
3. Give the project a name and give it a number.

4. If you would want the project to be moved to the top of the list, you should "**pin**" it (optional)
5. To launch the project, just double-click on its icon.

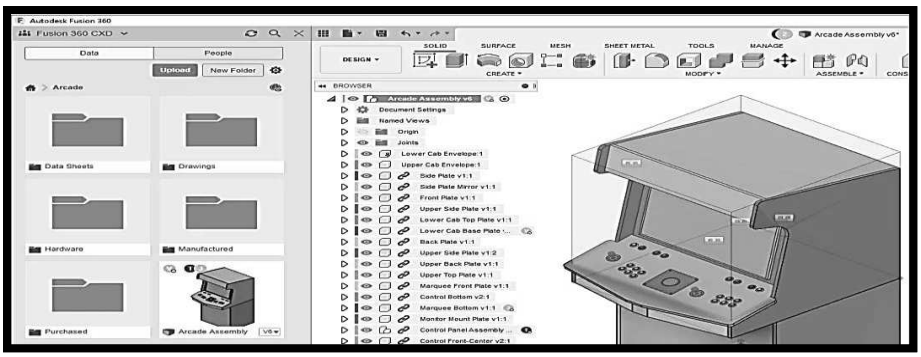
When you use the Data Panel to establish a new project, the Project Type setting will automatically be set to **Closed**. This ensures that the project can only be accessed by members of the team that you have invited to participate. During the process of creating a project in Fusion Team, you will have the opportunity to pick the **Project Type** and select a **Project Avatar**.



You can submit any data linked with the design to the project, and you can also invite other members of the project to participate. To launch Fusion Team and access the project, you may alternatively choose the **Open Details on Web** button from the toolbar. After that, you'll be able to use a web browser to control the project's settings, members, and content.

How to Create a Folder Structure

The newly created project does not have any kind of folder structure, to begin with. You can save a fresh design and then instantly begin working on more manageable jobs. On the other hand, if you're working on a more involved project with several people, you should devise a plan right from the start that will help you keep both your team and the project organized.



1. While the project is still open, go to the Data Panel and select the **New Folder** button. This will cause the creation of a new folder inside the project.
2. Give the folder a name and click the **Save** button.
3. Repeat the steps for every other folder and subfolder that you need.

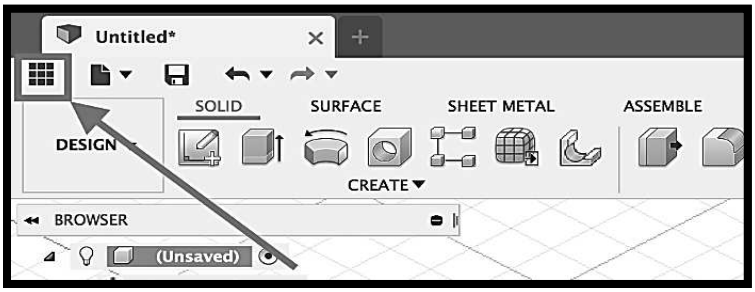
For the sake of this illustration of an arcade, you may wish to design a:

- **The manufactured** folder will include the components that you will develop and build in-house.
- **Purchased** folder for components, such as controllers, that you will buy off the shelf, such as in this case.
- **Drawing** folder to be used to store any designs that will be sent to third-party manufacturers
- **Data Sheets** folder that will include any specifications or data sheets connected with components in the design.
- **Hardware** folders that you may need, such as hinges, fasteners, and threaded inserts

Uploading Existing Files in a Project

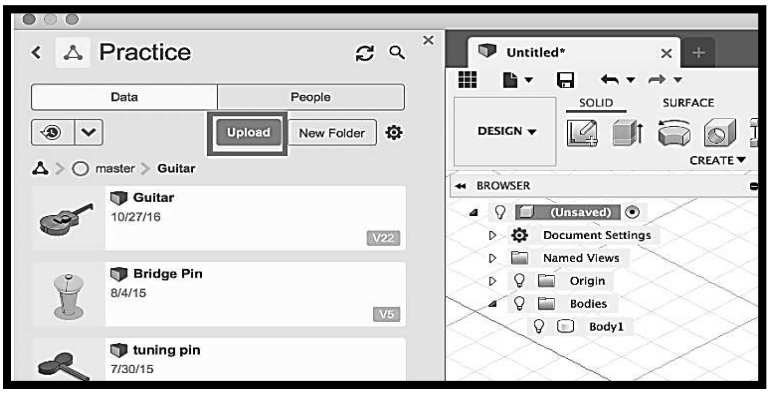
To upload a file to the Data Panel, do the following:

1. If the Data Panel is not already open, you may expand it by clicking the grid symbol in the top left corner of the screen.



2. Launch the project where the design will be stored after the Data Panel has been enlarged and then click the **Save** button.

3. Select the file to upload by clicking the icon as shown below;



4. In the Upload dialog box, click the "**Select Files**" button, as shown in the figure below;

5. On the personal computer, pick the directory where files may be uploaded, then open the file you want to upload by clicking the "**Open**" button.
6. After clicking "**Open**" to upload the design to the cloud, the design ought to be listed on the following screen.
7. After the file has been successfully transferred to the cloud, the status should have transitioned from a progress bar to the word "**Complete**."

Organize and find projects

All Open, Closed, and Secret projects that you are a part of are shown on the Projects tab in Fusion Team. The projects are organized alphabetically.

Projects can be pinned for organization. Projects can be located by:

- Browsing the projects list on the Projects page
- Sorting
- Filtering

Pin projects

You may easily identify projects that you are interested in or those you are working on by pinning them. You can pin both projects you are a part of and all Open projects.

Fusion 360

1. Launch the Data Panel first.
2. Hover your mouse over the project in the project list.
3. Select the **Project Pin Icon**.

A new filter called PINNED will become accessible on the Projects page when you pin a project.

In Fusion Team

- Choose the ALL filter on the Projects page.
- Hover your mouse over the project in the project list.
- Select the **Project Pin Icon**.

Unpin a Project

You may unpin a project if you no longer want it to appear in the PINNED filter.

- Choose the **PINNED** filter from the Projects page.
- Hover your mouse over the project in the project list.
- Select the **Project Unpin** icon.

Wherever the project is displayed, the Unpin option is accessible. You may do this, for instance, by hovering over the project on the page for all projects, in OWNED BY ME, or SHARED WITH ME.

Find content in the content

From the Project Home page in Fusion Team, you can access all the material in your project.

By default, a project's material is shown in the following order:

- Folders are displayed in alphabetical order
- Files displayed in alphabetical order

In your projects, you may modify how the material is displayed. To switch between the List view and Grid view choices, click the corresponding icons in the upper right corner of the Project content area.

Search for content

You may search for files by utilizing letters or phrases that are present in either the file names or the content of the files themselves. You may search for certain letters, characters, or phrases by clicking the search button that is located on the top right of the navigation bar and entering the information that you are looking for.

Filter projects

By default, the following filters are applied to the projects to organize them: **ALL, OWNED BY ME,** and **SHARED WITH ME.** Click the **OWNED BY ME** link to see and access all of the projects that you have established, and click the SHARED WITH ME link to see and access all of the projects to which you have been invited.

If you choose a filter, the default view will switch to the filter you choose the next time you sign in, regardless of whether you changed it or not. Using the Filter box that is located at the bottom of the Data Panel in the Fusion 360 app, you can filter the project list. The filter looks at the names of projects and the descriptions of those projects. As soon as you type a character into the Filter box, the filtering process will begin. Filtering may be done with anything from a single character up to multiple words. Either choose the Clear button that is located at the very end of the Filter field or erase the text that is included inside the filter. If the filter requirements are not met by any project or description, then all of the projects will be hidden.

Organize the contents and the projects.

You can arrange projects in descending order based on the project name (**Name**), the creator (**Owned by**), and the creation date (**Created On**). Simply click the column heading in the header row located directly above the list of projects to sort the projects in the list.

You can organize the project's files and folders according to the following criteria:

- File name (Name)
- Who was the uploader? (Owner)
- File type (Type)
- File size (size)
- The date of the most recent update (Last updated)

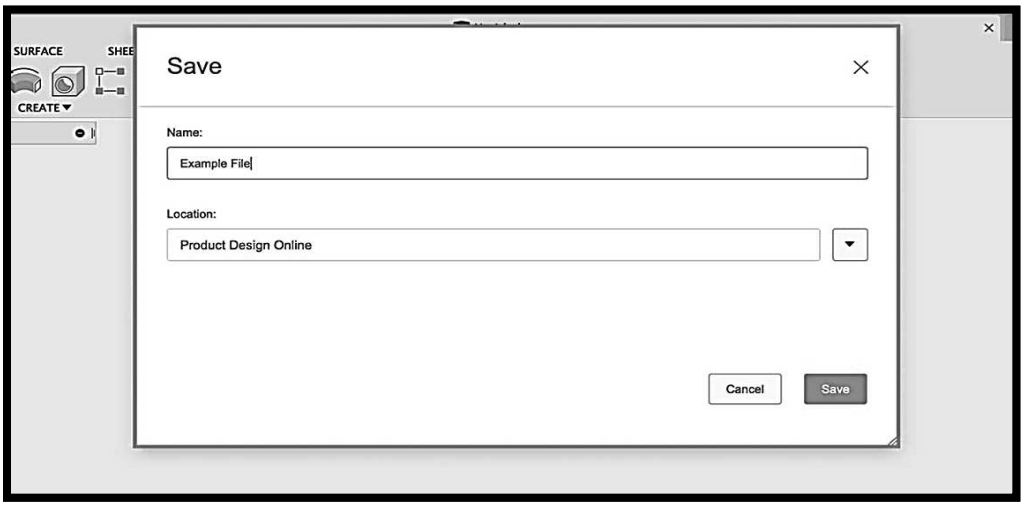
Take note that files and folders are organized in different ways. If you choose to sort by Owner, for instance, folders will be shown in alphabetical order according to the name of their respective owners (**ascending or descending**). Additionally, files will be arranged according to the owner's name in alphabetical order (**ascending or descending**). Folders will always come before files in the directory tree. These criteria are shown in the List view on the row that serves as the heading for the list of items. To sort the material, click on the header of any of the columns.

To sort the data, choose a criterion from the drop-down menu located in the Grid view.

Saving a Design File

Click the **save (disk)** icon that is located in the upper-left corner of the toolbar to save a design file. You may also use the keyboard shortcut **CMD + S (Mac OS) or CTRL + S** (Windows) (Mac).

When you save a design file for the first time, you will be required to provide the file name as well as the location where the file is going to be saved. Later on, using the data panel, you will be able to alter either the location or the name.



After you have successfully saved the file, the save icon will be disabled until the file is modified.

Exporting a Design to Other CAD Formats

Native vs. Neutral File Formats

If you have ever used Fusion 360 to open a generic CAD file, you have probably seen uneven results. On the other hand, one could not show up at all, while the other might be made up of hundreds of surface bodies. And why did some of the intricate details from the first model get lost in the translation? Translation problems can arise whenever a CAD file of any sort is imported or exported. The flexibility of design files to be exported and imported into a variety of tools raises the risk that quality may be compromised. Keeping

this in mind, it is essential to choose the appropriate file format to ensure that the quality of your design is maintained even when it is passed from one engineer to another. The file formats used by CAD programs may be divided into two distinct categories: native and neutral. The file formats that are native to a tool are inherently compatible with that tool. This indicates that the creator of the program controls the technology behind the file format, and they can make any modifications to it that they see fit. The creation of neutral file formats often occurs inside a standards-setting group of some type. This group takes into account not only one tool but also how their file format will be utilized across the board in terms of CAD software. The intention is to make it simple for any CAD tool to switch between using any of these file formats.

Keeping all of this in mind, let's discuss some of the most common neutral file formats that you'll want to think about using when exporting your next design:

STEP

The **International Organization for Standardization** (ISO) governs one of the most used neutral file formats, which is known as the **Standard for Exchange of Product Data** (STEP). Probably, you are already familiar with the STEP file format if you have dealt with almost any kind of 3D CAD.

This format, in comparison to other neutral forms, provides a variety of benefits, including the following:

- **More data**. Tolerance information is included in STEP files, which offer valuable data for applications requiring accurate machining.
- **More versatility**. The STEP format, which is defined by ISO standards and has been updated throughout the years to serve a wide variety of specialized engineering specialties, may be found here.
- **More intelligence**. STEP files reference each unique component included inside a design rather than reproducing the same part an arbitrary number of times; as a result, this format is ideal for use with assemblies.

IGES

Before the release of STEP, the dominant CAD file format was known as the **Initial Graphics Exchange Specification** (IGES). STEP was developed to replace IGES. This particular file format provides geometry data for a model, but it does not include any information about the connection that pieces have with one another inside an assembly. Additionally, it does not provide built-in support for solid modeling. There is a possibility that IGES files are still in circulation; nevertheless, we do not advise that you use these files to translate your contemporary CAD drawings. As of the time of this writing, the IGES standard will not be undergoing any more revisions in the foreseeable future.

Parasolid

The **Parasolid** file format is owned by Siemens, and any other corporation that is willing to fork out the necessary cash may get a license to use it. This format has a CAD kernel, which assures that it is compatible with a wide range of different tools.

Among the many modeling approaches that may be used with Parasolid are the following:

- Solid modeling
- Modeling of freeform surfaces and sheets
- Support for the rendering of graphics, including tessellation

DXF

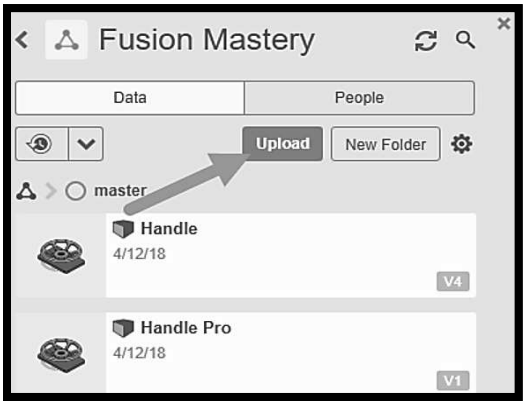
The **Drawing Exchange Format**, known as DXF, is AutoCAD's default file format for 2D drawings. We have decided to include it here because AutoCAD is one of the most popular CAD programs, and practically all CAD programs support DXF. This format is wonderful for bridging the gap between old and modern applications; but it does not support the most recent technology such as solid modeling.

STL

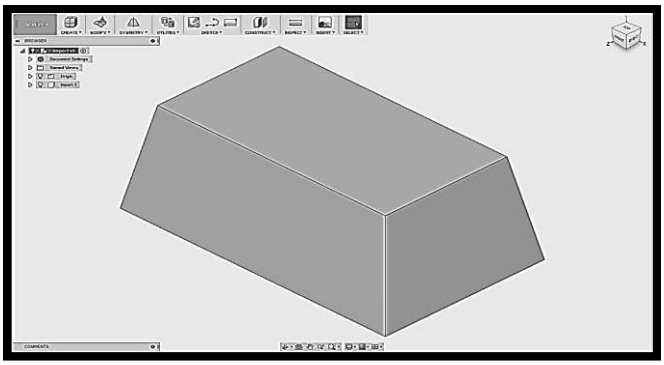
The **Standard Tessellation Language** (STL) file format is used extensively in 3D printing, scanning, and even certain CAM software programs. This model is represented in this file format as a pure triangular mesh; however, none of the object's extensive parametric data is included in the representation. Because of these constraints, STL works well for 3D printing, but it is not recommended for use with 3D CAD tools.

Importing a CAD File in Fusion 360

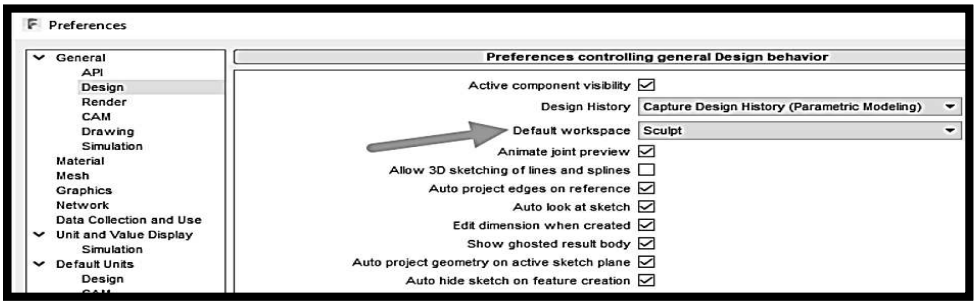
To get started, you'll need a CAD file to practice importing into so you can hone your abilities. The process of uploading a Fusion 360 design and importing standard CAD files are functionally identical. You will need to start by bringing up the Data panel, selecting a project from the drop-down menu, and then clicking the Upload button.



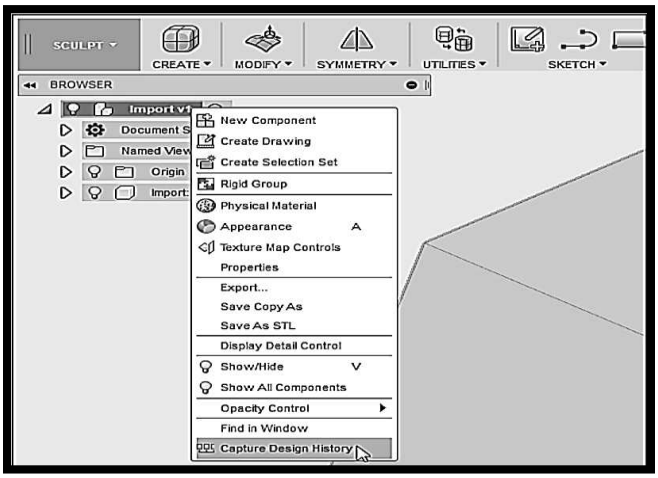
You may now choose the IGES file you just downloaded and then click the option labeled Upload from this screen. After the file has been completely uploaded, you may then go ahead and shut the Job Status window. **When you open the example IGES file in your canvas, you should see a model that has a fancy appearance similar to the one below:**



When it comes to importing generic CAD files like this one, there are a few important factors to keep in mind. To begin, the Sculpt workspace will be selected as the default when you have finished importing your file. **This may be altered in the settings of Fusion 360, as seen in the following example:**

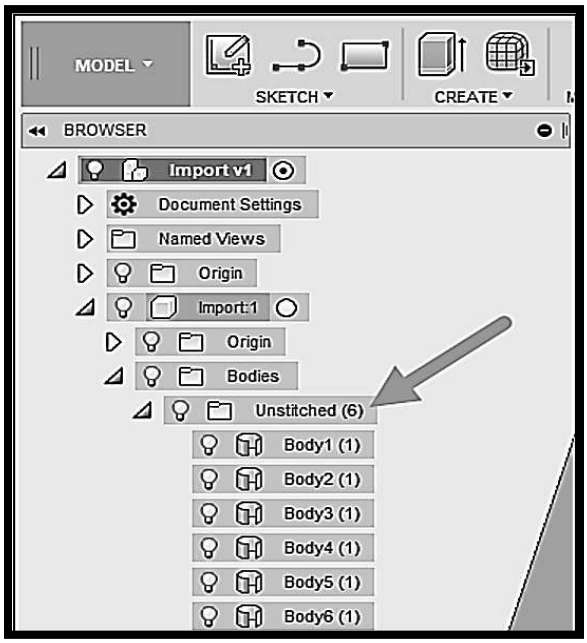


You'll also note that there isn't a design timeline at the bottom of the canvas in this particular file, which brings us to our second point. To activate this feature, right-click the file name while in the Browser, and then pick the **Capture Design History** option from the context menu.

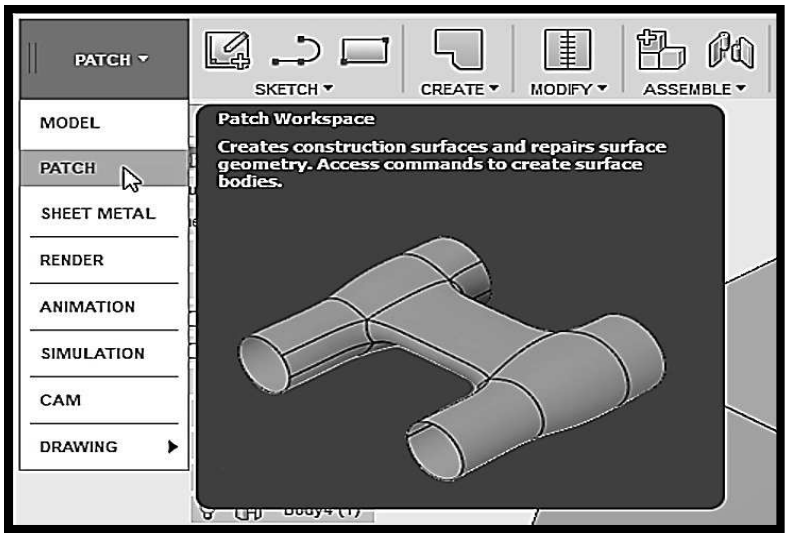


When you choose the plus sign on your newly added timeline, you will notice two entries: the first is for the original import, and the second is for something that is referred to as **Base Feature 1**. When a solid or surface body is imported, it is given a name like this, which is essentially a generic name.

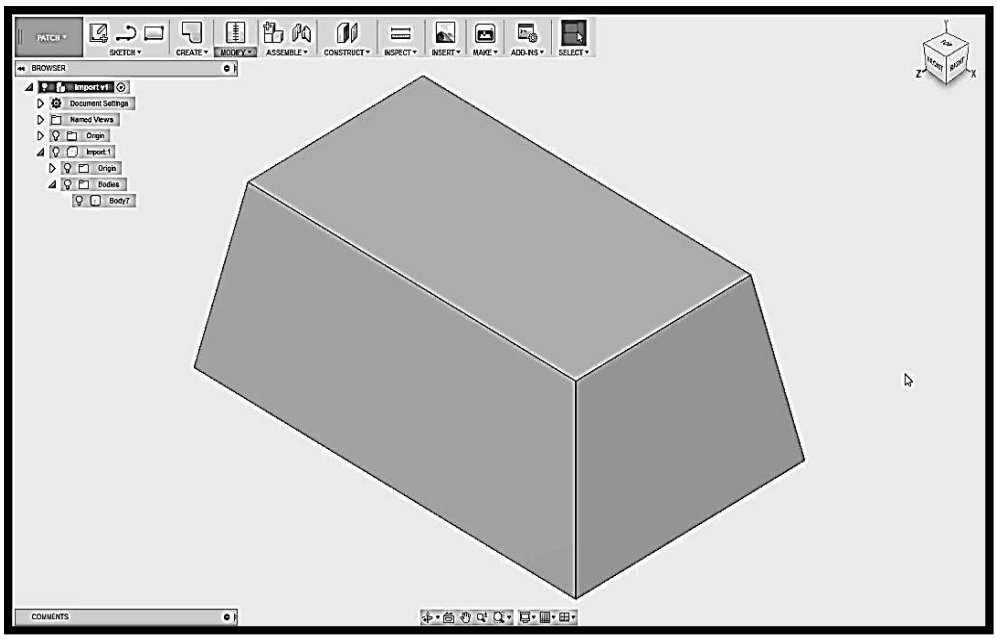
Last but not least, Fusion may recognize some imported objects as a collection of separate bodies rather than a single solid body. First, open the **Import: 1** folder on your browser, and then expand the Bodies folder. You will notice a folder labeled "**Unstitched**" that has six separate bodies within.



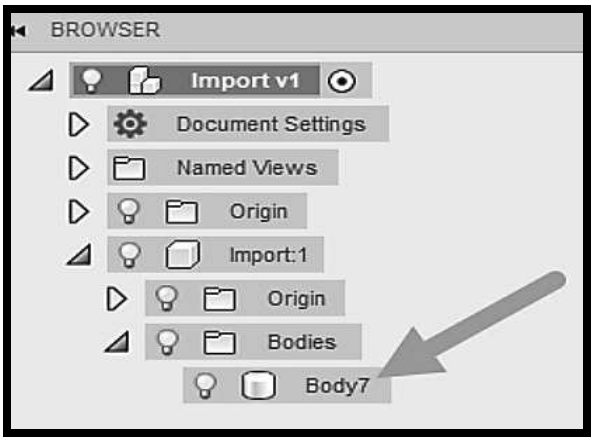
What led to this occurrence? When you want to save an IGES file, you will often be given the choice to either save it as a surface or as a collection of surface bodies. Once we have switched to the Patch workspace, we will be able to easily sew these bodies together.



Using the shift key and the left mouse button, select all of the bodies that are found in the **unstitched folder** inside the Patch workspace. When you have all of them chosen, expand the Modify choices on the toolbar of the Patch workspace, and then pick the **Stitch** option. In the Stitch dialog, we are going to leave all of the settings as they are, and then click the **OK** button.



Once the stitch is finished, you should check out the results on your browser. These six unstitched bodies need to have been combined into a single Body 7 at this point.



At this stage, our IGS model has been loaded into Fusion 360 without incident and the components have been successfully joined together. You are now working on V2; just be sure to save the file and give it a description that reflects the change.

Exporting a CAD File in Fusion 360

The process of importing a standard CAD file has just been covered; the next step is to learn how to export the same file to a different CAD user.

There are two possible courses of action to investigate:

Exporting from Fusion 360

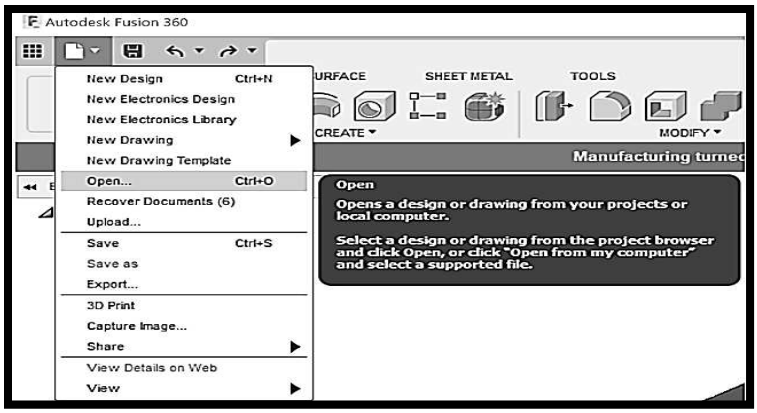
Choose **Export** from the File menu while your IGES file is still open. The Export window will appear, providing you with the choice to save the file to either the Fusion cloud or your local computer. It will also provide you with a few file-type options to choose from.

You may have seen that the list of supported file formats in the dropdown is quite limited. If the format you want to use isn't on the list, it's time to step into Fusion Team.

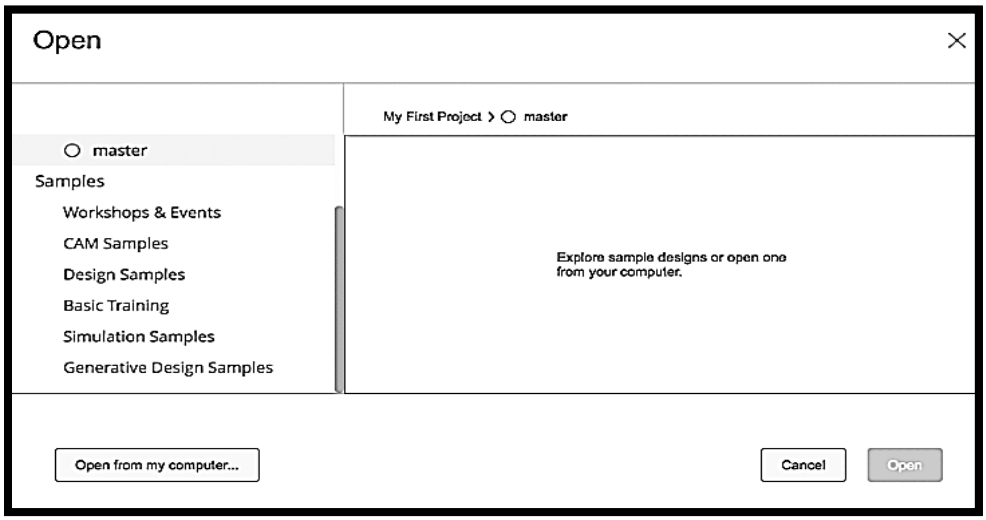
Opening an Existing Design File

The file may be an existing Fusion 360 design that has been saved in the past, or it can be a file that was saved locally on your computer by following the procedures below:

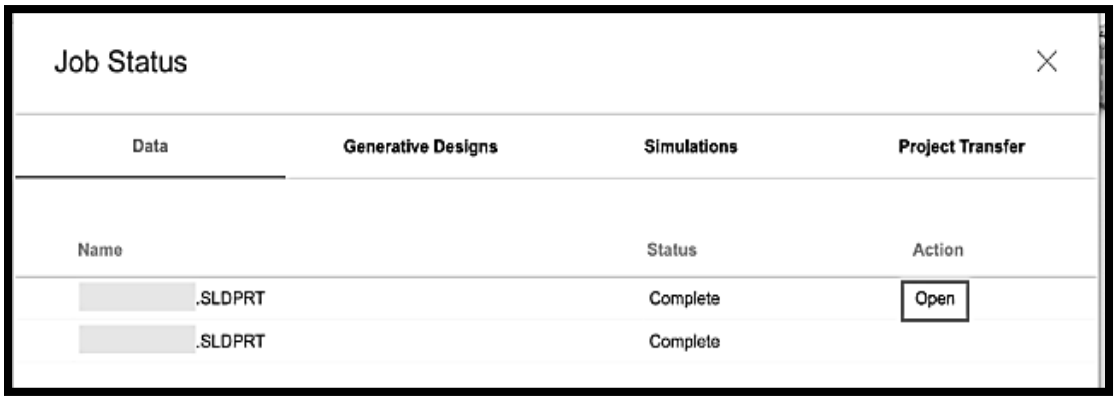
1. Click **File**.
2. Click **Open**.



3. A dialog window labeled **Open** will then open, displaying your cloud data.
4. To access a file that is listed in your cloud data, go to the file's location, and then click the Open button.
5. Select "**Open from my computer**" to load a file that has been stored locally on your computer.

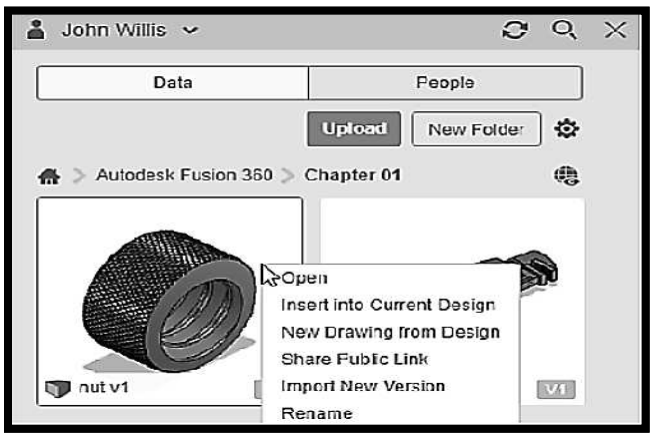


6. Once you have located the file, click the **Open** button. For a list of the many sorts of files that can be read by Fusion 360.
7. When the translation of the file in the Cloud is finished, you may open it in Fusion 360 by clicking the "**Open**" button in the Job Status box. This will bring up the file.



Opening an Existing File from the Data Panel

Simply select the Show Data Panel option on the Application Bar. This will allow you to access an existing design file of a project from inside the Data Panel. The Data Panel is brought into play. The next step is to go to the location of the file that needs to be opened, then double-click on the file itself. Fusion 360 will now open the specified file in its workspace. Another way is to right-click on the file that needs to be opened, then click on the **Open** option that appears in the shortcut menu that pops up.



When you open a design file in Fusion 360, the program will automatically open the most recent version of that file. Nevertheless, you also have the option of opening the file in an earlier version. To achieve this, choose the design thumbnail in the Data Panel and then click on the Version icon that is located in the bottom right-hand corner of the thumbnail. The Data Panel contains representations of all the different

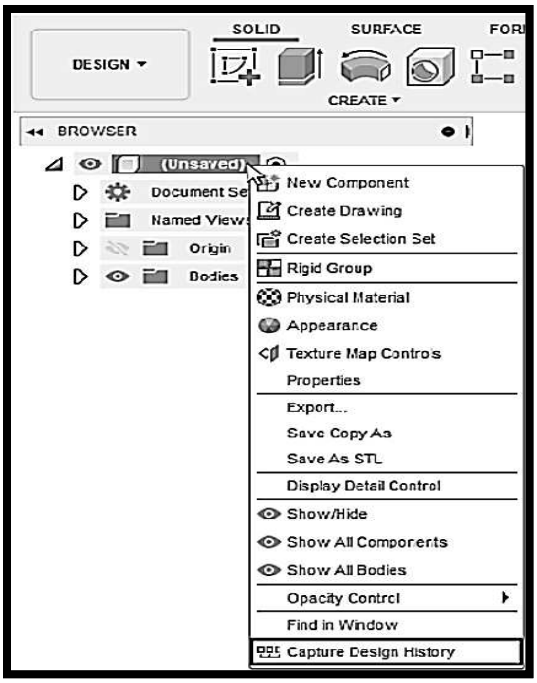
versions of the respective design file. Move the mouse over the version of the file that you want to open and leave it there. In its immediate vicinity are the buttons labeled **Promote and Open.** When you click the **Promote** button, the older version of the file will be upgraded to the most recent version of the file. The Open button is what you need to click to open the earlier version of the file. To access the file, choose the **Open** option. Fusion 360 will launch with the version of the file that you choose already open. If the Data Panel does not display all versions of the files that have been chosen, you may make them show by selecting the option to Show all versions inside the Data Panel.

Opening an Existing File by using the Open tool

To use the Open tool to access an existing design file associated with a project, first, choose **File** from the menu located in the Application Bar, and then select **Open** from the menu that appears. The dialog window labeled Open will now appear. In this dialog box, you need to choose the project by clicking on its name, which is located on the left panel of the dialog box. The right panel of the dialog box displays all of the subfolders and files that are associated with the currently chosen project. Choose the necessary design file to open from the list located in the right panel of the dialog box. Note that if the design file that is going to be opened is saved in a subfolder of the project that is now chosen, then you will need to double-click on the subfolder to see the files that are stored inside of it. After choosing the file that contains the design, click the **Open** button that is located in the dialog box. The application will open the specified file.

Opening an Existing File from the Local Computer

You also have the option of opening a previously stored file on your local computer, such as a Fusion file **(*.f3d), IGES file (*.iges; *.igs), SAT file (*.sat), SMT file (*.smt), STEP file (*.step; *.stp),** etc. To do this, bring up the Open dialog box, and then choose the option to **Open from My Computer** from the menu that appears. There is a new instance of the Open dialog box appearing. Use the "**Browse**" button in this dialog box to go to the location of the file that needs to be opened. You will be able to open files created in Alias, AutoCAD DWG, Autodesk Fusion 360, Autodesk Inventor, Catia V5, NX, IGES, and STEP, amongst other file formats. Following the selection of the necessary file, the **Open** button will become active in the dialog box. When you click on the Job State window, you will see the current status of the file shown in the Status column. Click the **Open** option in the Action column of the dialog box when the status of the file displays as Complete in the Status column of the dialog box. This will allow you to access the file. Fusion 360 will now open the specified file in its workspace. When you access an already-created file, you will have the ability to update it by adding new features. It is advised that before you add the new features, you switch on the process of recording design history in the Timeline for the newly added features. This should be done before you add the new features. To do this, right-click on the name of the file (the top browser node) in the **BROWSER**, and once the shortcut menu opens, choose the tool titled "**Capture Design History**" from the list of available options. In the same manner, if you do not want to record the design history for a particular design, you may use the tool that allows you to do so by selecting it from the shortcut menu that displays.

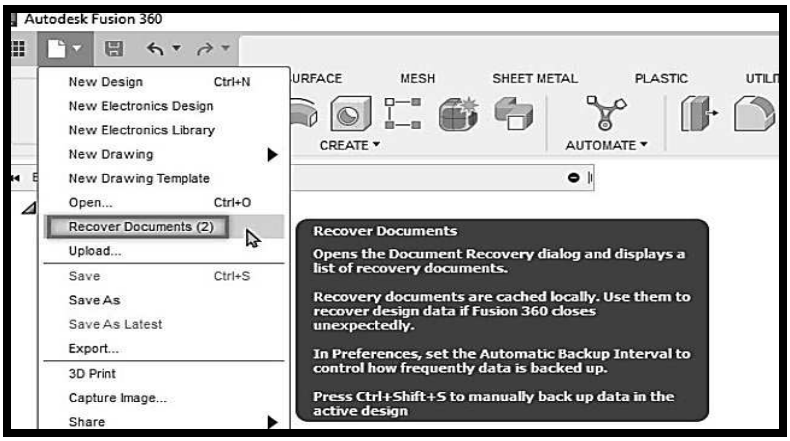


Working in the Offline Mode

When you are not connected to the Internet, when your Internet connection is lost, when an unexpected outage is detected, or when the service is being maintained, Fusion 360 will switch to the offline mode automatically. Nevertheless, Fusion 360 enables you to continue working on your ideas even when you are not connected to the online platform. In addition, you can manually toggle between the online and offline modes. To check the current status of your job, go to the top right corner of the screen and click on the button labeled "**Job Status**." The flyout for the Job Status will show. To begin working online, choose the icon provided in this flyer. When this happens, the offline mode is enabled, and an icon denoting "**Working Offline**" shows in the flyout.

Recovering Unsaved Data

If you want to identify a file that can be recovered, you may try using one or more of the following solutions: To examine the designs that were recovered from the file, use the Recover Documents command.

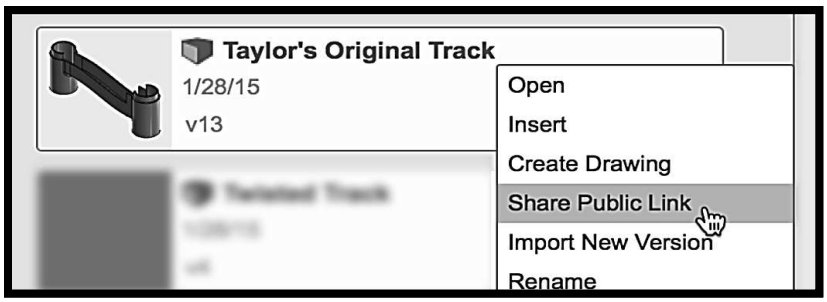


Use the arrow keys or the mouse to go to the next place. Make use of the upload command to successfully upload the necessary ***.f3d file**. Please take note that the files and folders in these directories are hidden. By default, backups will occur every 5 minutes (although this interval can be changed in the Fusion preferences). While you are working on the document, there is a chance that the save will be delayed until after the current operation has been finished. For instance, an automatic save will not begin if there is a dialog box for the Extrude command that is now active.

Sharing a Design

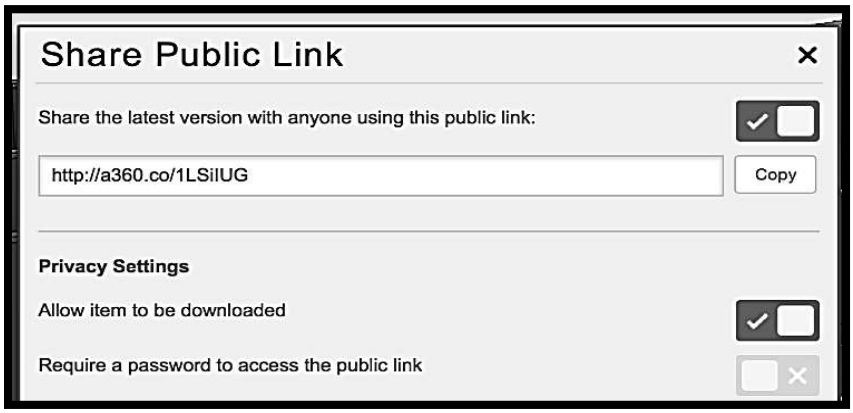
Sharing Design Using a Link

This public connection may be generated in one of two different places: either from inside Fusion 360 itself or from A360 within a web browser. Launch Fusion 360, open the Data Panel, and go to the location of the Design you want to share. Use the context menu to choose **Share Public Link** after right-clicking on the design.

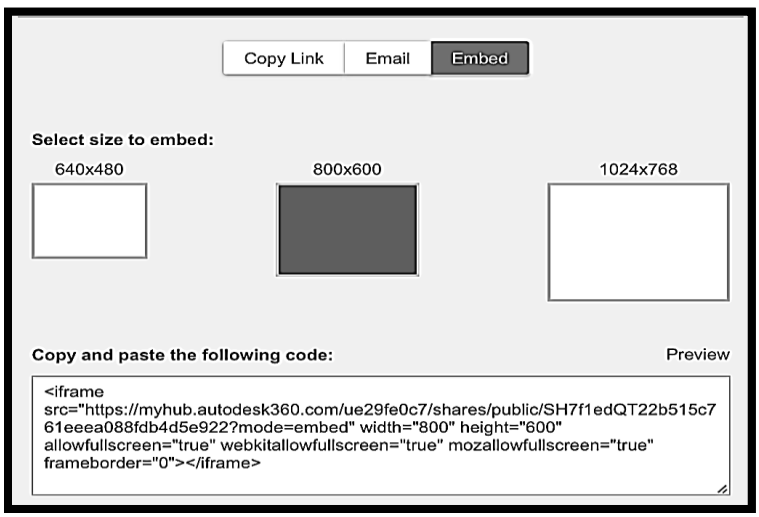


To make the public link active, you will need to tick the first box that appears in the following dialogue window. After that, you'll be able to choose whether or not the Design may be downloaded, as well as whether or not a password is needed to access it. When you make the Design accessible for download, you

are not only posting the.f3d file so that it may be downloaded by others, but you are also making it possible for anybody else to get the particular file format that they need. Instead of manually submitting files with extensions such as.f3d, STL, and.STP, let the cloud take care of this translation for you instead! You may start sharing with anybody you want after you have copied the link that is located at the top of the dialogue box. The public link will, for your convenience, continue to lead to the most recent version of your Design even as you make new versions of it in Fusion 360.

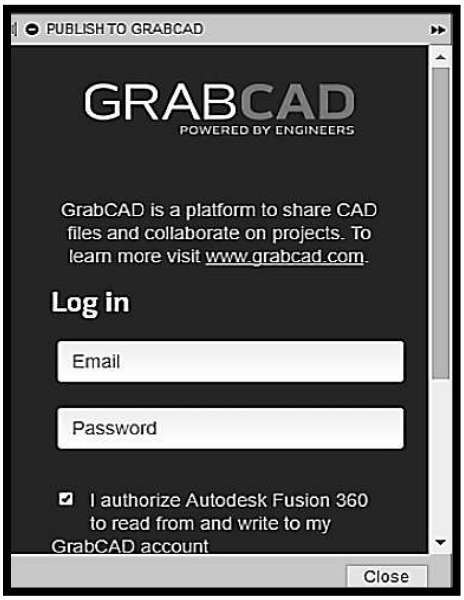


The use of A360 inside a web browser constitutes the second method for producing a public connection. Open the **A360 viewer** and browse the design inside A360 that you want to share once you have done so. There will be a **Share** button at the very top of the screen, and clicking it will bring up the identical dialogue that can be seen in the actual product. In conclusion, the A360 viewer enables users to produce HTML embed code, which can then be included in any website of their choosing to add a 3D viewer that does not need the use of a plugin. To do this, choose the share dialogue box and look for the Embed option. The only thing left to do is choose the size of the viewer you want, copy the HTML code, and then paste it into the HTML code of your website!



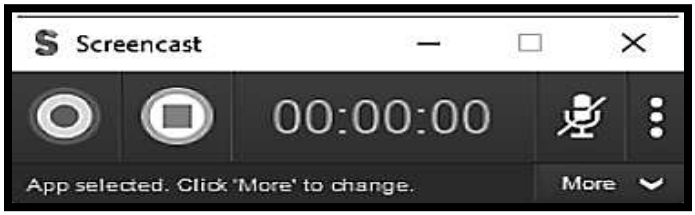
Sharing Design to GrabCAD

To publish or share your design on GrabCAD (www.grabcad.com), go to the **File** menu and choose **Share** before selecting **GrabCAD** from the drop-down menu. The dialog box labeled **PUBLISH TO GRABCAD** will now display. You may publish your design to all registered members of GrabCAD by logging in to your GrabCAD account via this window and then selecting the Publish button.

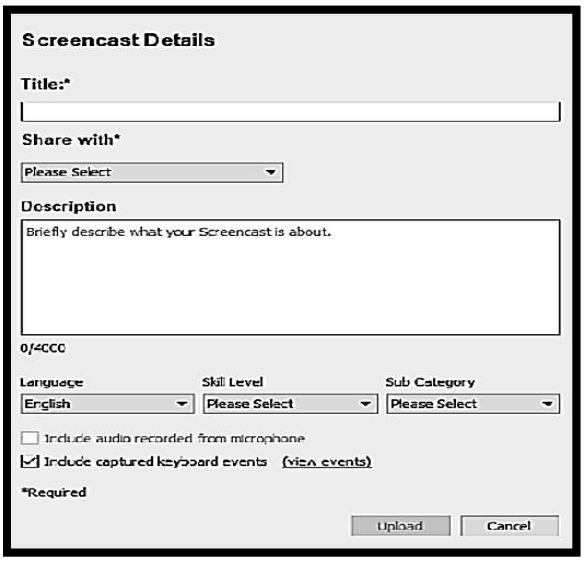


Sharing Design by Recording Screen

Using the Screencast Recorder, which is included in Fusion 360, you can record your screen and produce a movie of your design, which you can then send to anybody. To accomplish this, go to the application bar and click on the File menu, then choose Share, and then select Launch **Screencast Recorder**. The Screencast Recorder program is going to be downloaded; thus, the Autodesk website is going to be viewed. Get the Screencast Recorder program by downloading and installing it. After the Screencast Recorder application has been installed, you can open it by double-clicking the Screencast Recorder icon that is located on your desktop. The screencast window will now open.



After that, all you need to do to begin recording your screen is click the **Record** button located in this window. When you have finished recording all of the steps of your design workflow, you can end the recording by clicking the **Stop** button in the Screencast box. The preview window for the screencast recording is shown. To save and upload your changes, click the button located in this window. A dialogue window labeled Screencast Details is shown.



In this dialogue box, you will need to provide the necessary data, such as the title name, and description of the video. After that, in the Share with drop-down box, choose the option that corresponds to what you need, and then click the **Upload** button. The video will be uploaded and shared following the option that was chosen from the drop-down list located in the dialog box labeled Share with.

Invoking a Marking Menu

The Marking menu is a radial menu that gives you rapid access to commands that are used often.

- If you use your mouse's **right mouse button** anywhere in the canvas, the Marking menu will show around your cursor.
- To execute a command, you must first move the mouse in the direction of the command and then click anywhere inside the highlighted wedge.
- To close the Marking menu, either click anywhere outside of the menu, inside the center of the menu or hit the Esc key on your keyboard.

There are three distinct subsections inside the Marking menu:

- The radial menu on the first level
- The radial menu on the second level
- The context menu

First-level radial menu

The radial menu on the first level includes commands that are used often. There are eight predefined commands available in the radial menu of the first level in the Design workspace.

- Restate last command.
- PressPull
- Redo
- Hole
- Sketch
- Move/Copy
- Undo
- Delete

Workspaces, toolbar tabs, contextual environments, and active commands

The commands that are shown in the Marking menu are dynamic and alter depending on the workspace, the toolbar tab, the contextual environment, and the command that is now active.

Examples:

When you transition to the Surface tab in the Design workspace, the Hole command is replaced with the Patch command. This change takes effect immediately.

- The **Flange command** replaces the one that was there before when you switched to the Sheet Metal tab.
- The **Edit Form** and **Face commands** are shown when the Form contextual environment is selected.
- The **Generative Design workspace** shows instructions that guide you through the process of setting up generative research.
- The **Render workspace** presents instructions that are relevant to rendering, appearances, and textures, as well as scenarios.
- The **Animation workspace** provides access to commands that facilitate the creation of animations and their distribution.
- The **Simulation workspace** provides command prompts that guide you through the process of establishing simulation studies.
- The **repeat, undo, redo, and import functions** are not available in the **Manufacture workspace**; however, this workspace does provide a more detailed context menu for configuring and maintaining toolpaths.
- The **Drawing workspace** provides you with commands that assist you in documenting and annotating your design, in addition to tools for annotation that are accessible through the context menu.

Second-level radial menu

When you move the cursor over the bottom command in the radial menu of the first level, the radial menu of the second level appears around the pointer.

The second-level radial menu in Sketch has default access to eight different commands, which are as follows:

- Complete the Sketch
- 2-Point Rectangle
- Fit Point Spline
- Project
- Line
- Offset
- Sketch Dimension
- Center Diameter Circle

Your productivity may be substantially sped up and made more efficient if you access the Sketch commands via the second level of the radial menu. By moving the mouse above the up arrow, you may access the radial menu on the first level again.

Context menu

A context menu that includes the following items is displayed underneath the radial menu:

- Operation controls for navigation such as Pan, zoom, and Orbit
- Isolate/Unisolate
- A list of available workspaces will make switching between them much simpler.
- Saved shortcut keys

Choose the command you want to execute from the menu that appears when you right-click.

Gestures

You may start to use gestures to pick commands in the Marking menu after you have started to learn the placement of commonly used commands in the Marking menu. This allows you to select commands without having to bring up the whole Marking menu. Any command that is accessible through a radial menu may be executed using gestures. You may also use gestures to pick contextual instructions such as **OK** and **Cancel** while a command is active. This is particularly useful for the radial menu on the second level of the Sketch program. You can access the Sketch commands by right-clicking, holding down the mouse button, and dragging down, followed by dragging in the direction of where the command appears on the second-level radial menu of Sketch.

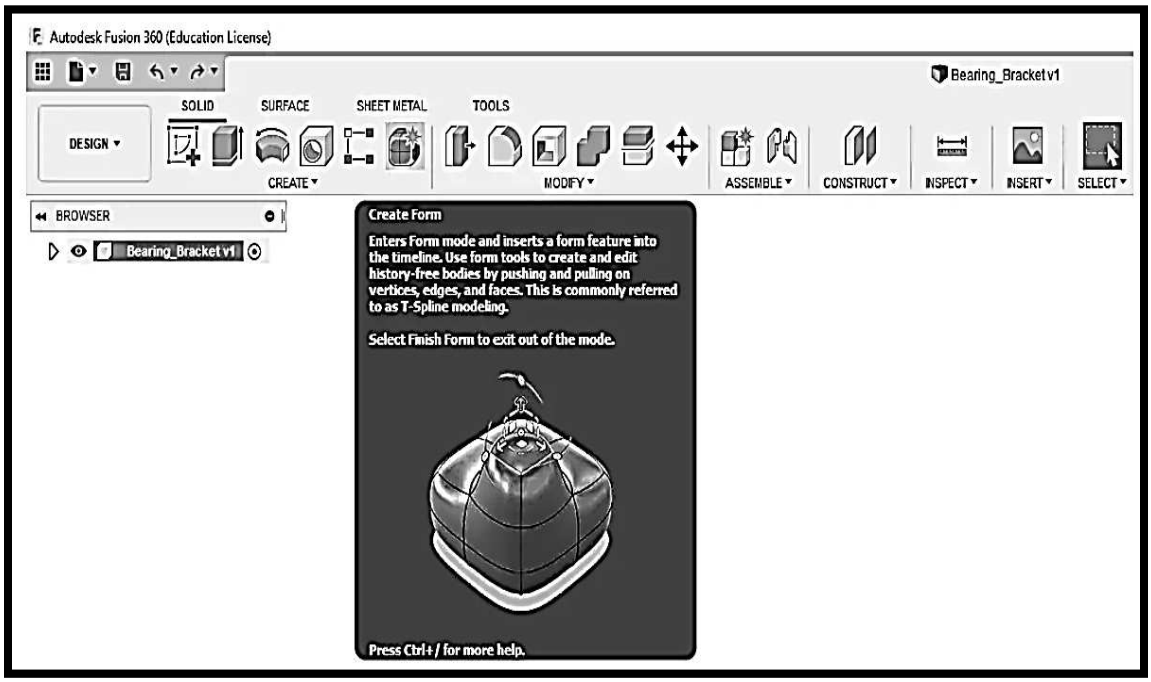
Examples:

- To create a 2-Point Rectangle, drag in the downward direction and then to the top right.
- To create a Fit Point Spline, drag in the form of an L.
- **Line**: Move in a downward direction.
- To adjust the offset, drag in the down and left directions.
- To complete the sketch, drag the cursor down, then halfway up.
- To do this action, within a command, right-click and hold, then swiftly drag to the right.

3D Printing

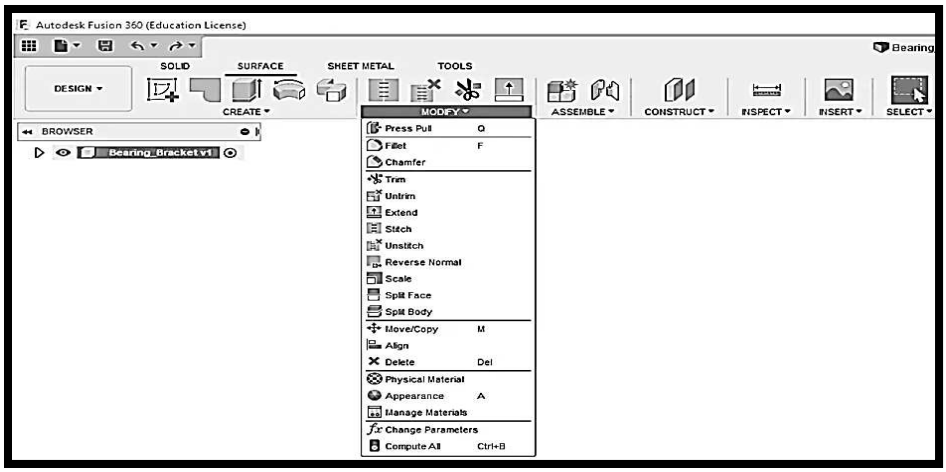
Form Feature

The form feature, which is represented by a purple cube, makes it possible to build intricate organic forms. It offers up a new workspace that comes equipped with a broad variety of tools for sculpting intricate designs. Using this function, one may easily create organic and creative models that can then be printed using 3D technology.



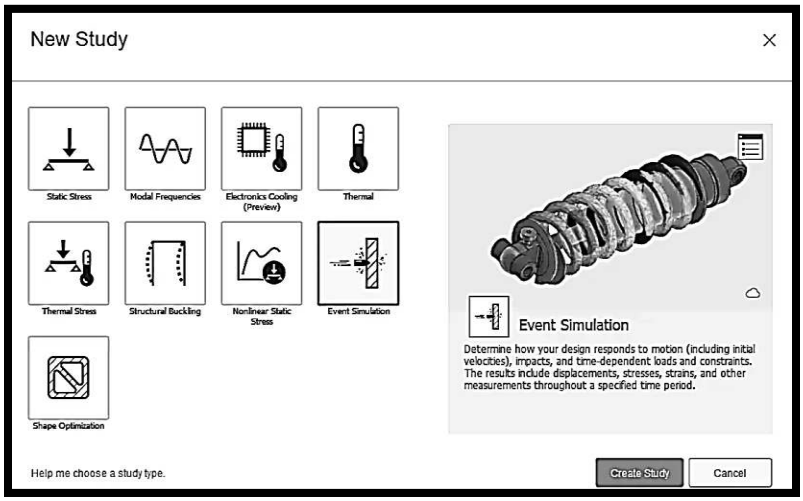
Surfaces

The user can fix models in preparation for 3D printing using surface tools. To reshape the component, the surfaces may be stitched closed, extruded, pulled, or pushed. A watertight model that does not have any holes in the shell may also be created with the assistance of the surfacing tools.



Event simulation

This tool helps model how your 3D print will react under time-dependent loads and velocities so that you may make any necessary adjustments before it is printed. For instance, snap-fit joints may be modeled to indicate what loads are encountered by the clip while it is being pressed closed. This provides a good understanding of where the weak areas are, allowing the design to be optimized more effectively.



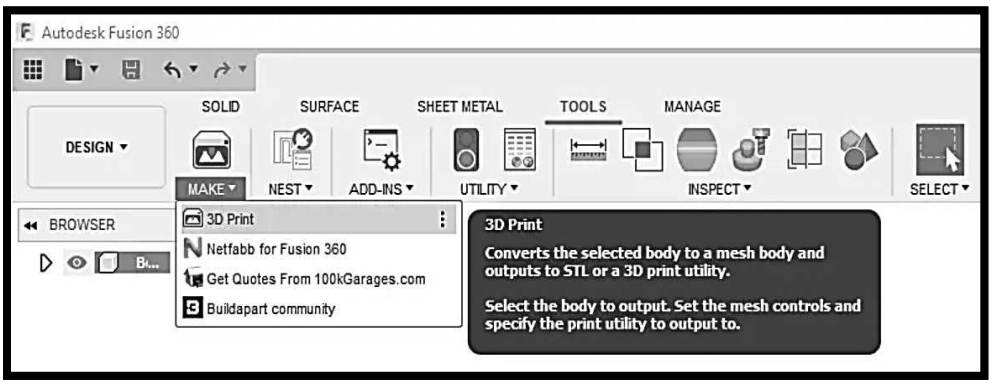
Non-Linear Stress

Certain 3D printing procedures, such as fused deposition modeling (FDM), result in the creation of components having non-linear material characteristics, which can only be modeled using a finite element analysis (FEA) tool that offers a non-linear study type. If the appropriate material data is input into Fusion

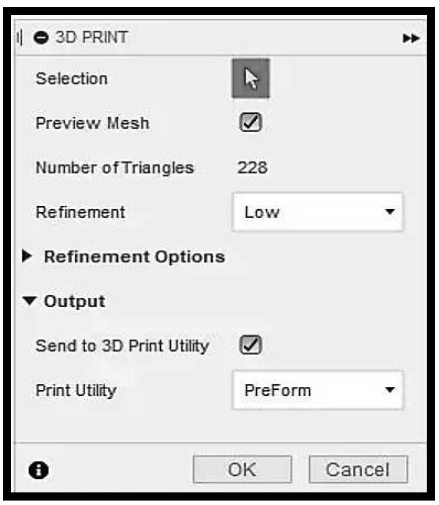
360, the extremely competent nonlinear study type that it offers can produce an accurate prediction of the stress that will be placed on a component.

3D Printing from Fusion 360

When the **Make** icon is clicked in the Design workspace, the 3D print menu appears. From this menu, you may make a variety of adjustments to the model to improve its readiness for printing, and then you can submit the model to a 3D print utility.



The following is a list of the many choices that may be selected from the menu:



Selection

The user is given the ability to pick the model to be 3D printed via the usage of this option.

Preview Mesh.

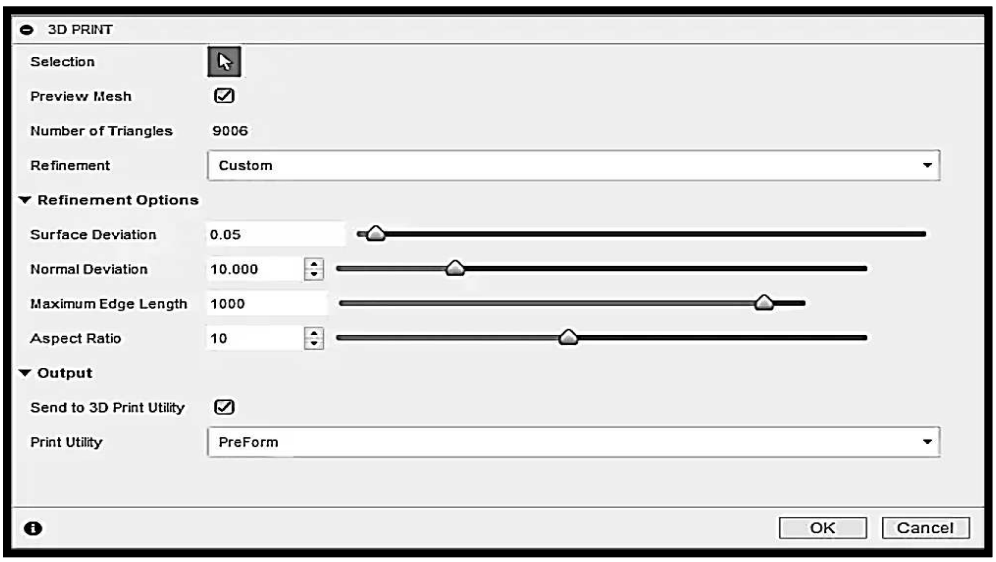
This checkbox displays the model's mesh, which is helpful for the user if the user wishes to see what impact modifications to the parameters have on the model.

Number of triangles

This reveals the total number of unique triangles that are used to construct the model. This number will rise when the fineness level is raised.

Refinement

With this option, you can choose between three different predefined levels of refinement: low, medium, and high. The overall number of triangles that are used in the model is determined by this. **In addition to this, there is a custom option that gives the user the ability to further customize the mesh depending on the following parameters:**



Exporting a Design in STL File Format for 3D Printing

Method #1: Exporting an Entire Design

Using the model Browser tree, which is located on the left-hand side of the screen, you can export a full project, which will include all of the bodies and components that make up the project. Simply right-click the top branch of the tree, which is where the project name is located, and then pick "**Save As Mesh**" from the drop-down option that appears.