

Overview

Fusion 360 is a cloud-based CAD/CAM tool for collaborative product development. The tools in Fusion enable exploration and iteration on product ideas and collaboration within a product development team.

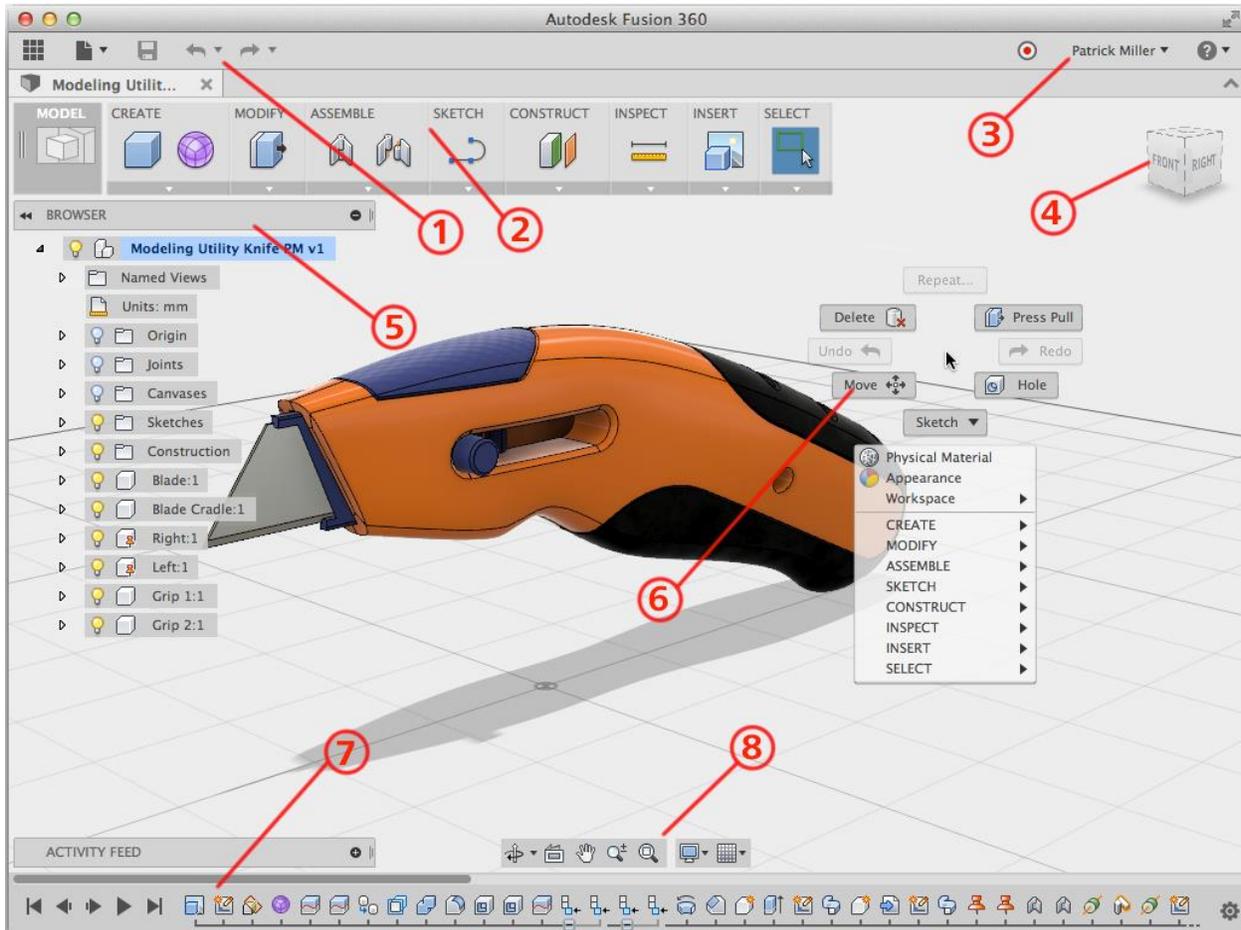
Fusion 360 enables fast and easy exploration of design ideas with an integrated concept to production toolset. Fusion lets you focus on the form, function, and fabrication of your products. Use the sculpting tools to explore form and modeling tools to create finishing features. These tools let you quickly iterate on design ideas. Once you have settled on a design, you can create assemblies to validate fit and motion in your design or create photo-realistic renderings to verify the appearance. Finally, you need to fabricate your design. Use the 3D print workflows to create a rapid prototype or the CAM workspace to create toolpaths to machine your components.

Fusion 360 also helps bring design teams together for collaborative product development. All your designs are stored in the cloud, which means you and your team always access the latest data. Fusion also tracks versions of your design as you work. You can use Autodesk A360 to view each version in your web browser and promote an old version to the current version. Finally, use Fusion and A360 to share your designs and track design activity. You can even provide controlled access to your designs without requiring an Autodesk ID.

Fusion 360 uses a hybrid environment that harnesses the power of the cloud when necessary and uses local resources when it makes sense and cloud resources. For example, your design data is stored on the cloud and renders amazing images every time you save a new version of your design. This happens in parallel while you are creating and editing designs local on your machine. This allows you to harness the power of your computer and the power of the cloud at the same time.

Throughout this course, you explore these areas of Fusion 360. This course will get you started designing with Fusion and help you understand how it can improve your design processes.

Main User Interface



1. Application bar:

-  Data Panel: Display or hide the data panel on the left of the interface.
-  File: Access file operations such as New Design, Save, Export, and 3D Print.
-  Save: Save an untitled design or save the changes to a design as a new version.
-  Undo/redo: Undo or redo operations.

2. Toolbar: Access commands in the toolbar.

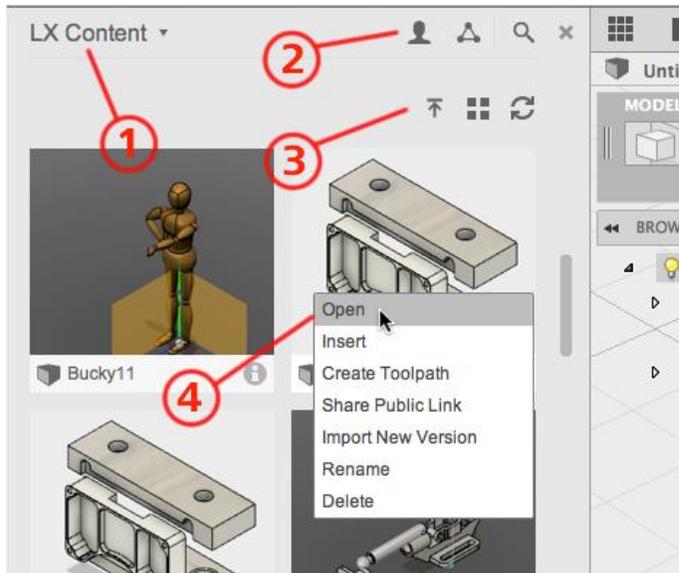
3. Profile and help:

- Profile name: Access preferences and your Autodesk profile.
-  Help: Access help, forums, and tutorials, what's new, and feedback.

4. ViewCube: Orbit the view and access orthographic and isometric views.

5. Browser: Lists objects in the design.
6. Marking menu: Another method to access commands. Right-click to display the marking menu.
7. Timeline: List the operations performed on a design if parametric modeling is active.
8. Navigation bar and display settings: The navigation bar contains commands to navigate the view. The display settings control the display of the design in the canvas.

Data Panel Interface



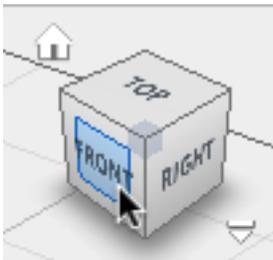
1. Project switcher: Select the active project.
2. Project tools
 -  Project Members: Invite members to the active project.
 -  Project details: Opens the active project in Autodesk A360 in your default internet browser.
 -  Search: Search the active project or all projects you have access to.
3. Data tools
 -  Upload: Upload files to Autodesk A360. Many CAD data types are supported as well as standard files such as documents, spreadsheets, and presentations.
 -  Data view: Select how data is displayed in the data panel.

-  Refresh: Refreshes data from Autodesk A360.
4. Thumbnails: Right-click a thumbnail to access commands for that specific design.

View Navigation

Commands

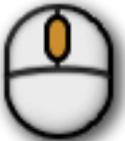
Use ViewCube to orbit the design in the canvas. Drag the ViewCube to perform a free orbit. Click faces and corners of the ViewCube to access standard orthographic and isometric views.



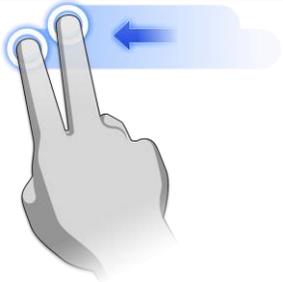
Use the commands in the Navigation bar to pan, zoom, and orbit the canvas. The menus on the right end control Display Settings and Layout Grid options.



Mouse

SCROLL		Scroll the middle mouse wheel to zoom in or zoom out.
HOLD		Click and hold the middle mouse button to pan the view.
SHIFT KEY +		Hold the SHIFT key and click and hold middle mouse button to orbit the view.

Mac Trackpad

	2 finger pinch to zoom out.
	2 finger spread to zoom in
	2 finger swipe to pan the view.
SHIFT + 	Hold SHIFT and 2 finger swipe to orbit the view.

Workspaces

Fusion 360 uses workspaces control the commands that are available and the type of data that is created. There are multiple workspaces available depending on the work you plan to perform.

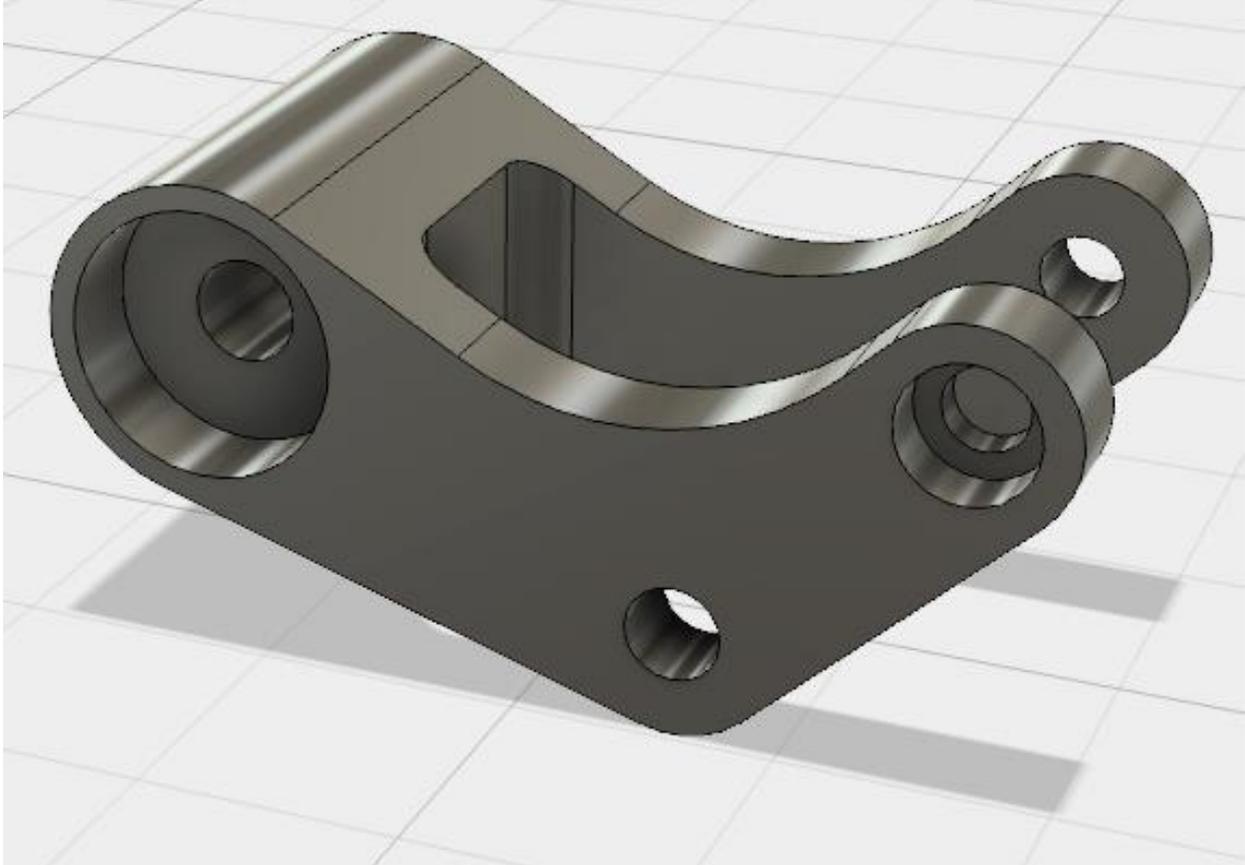
- Sculpt: create organic shapes by manipulating faces, edges, and vertices.
- Model: create solids with hard edges and flat faces.
- Patch: create open surfaces to stitch into solid bodies.
- Render: set up the environment and create photo-realistic renderings.
- CAM: create and simulate tool-paths then generate g code for subtractive manufacturing.
- Drawing: generate 2D manufacturing drawings.

You also have a drawing workspace for documentation, render workspace for creating photo-realistic renders, CAM workspace for creating toolpaths.

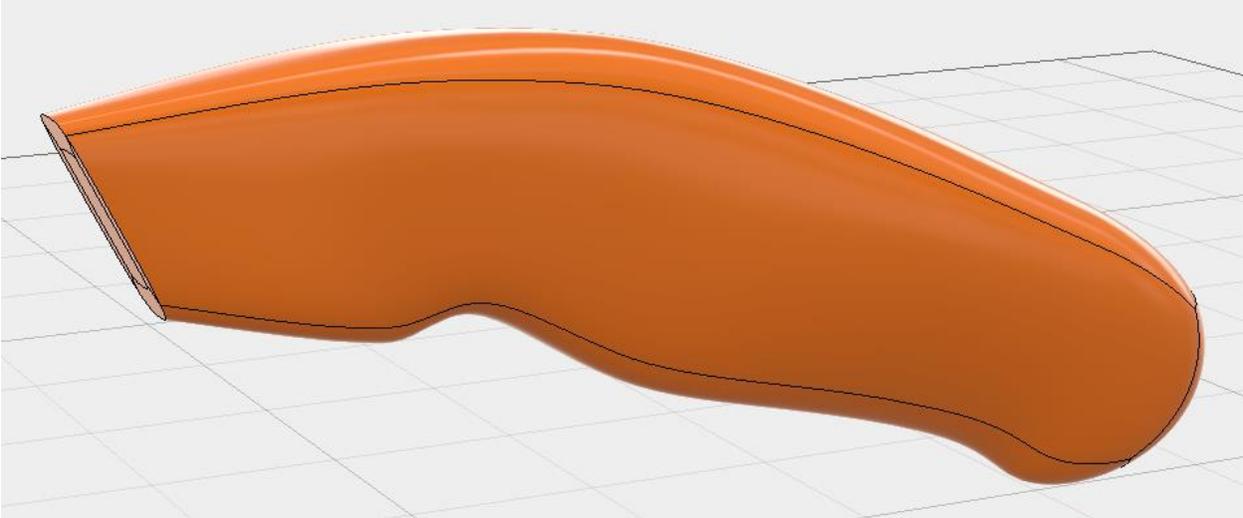
Autodesk Fusion 360: Introduction

It's obvious when to use some workspaces. If you need a 2D manufacturing drawing, you use the drawing workspace. What about model and sculpt? They are both used to create 3D designs so how do you choose to use one over the other?

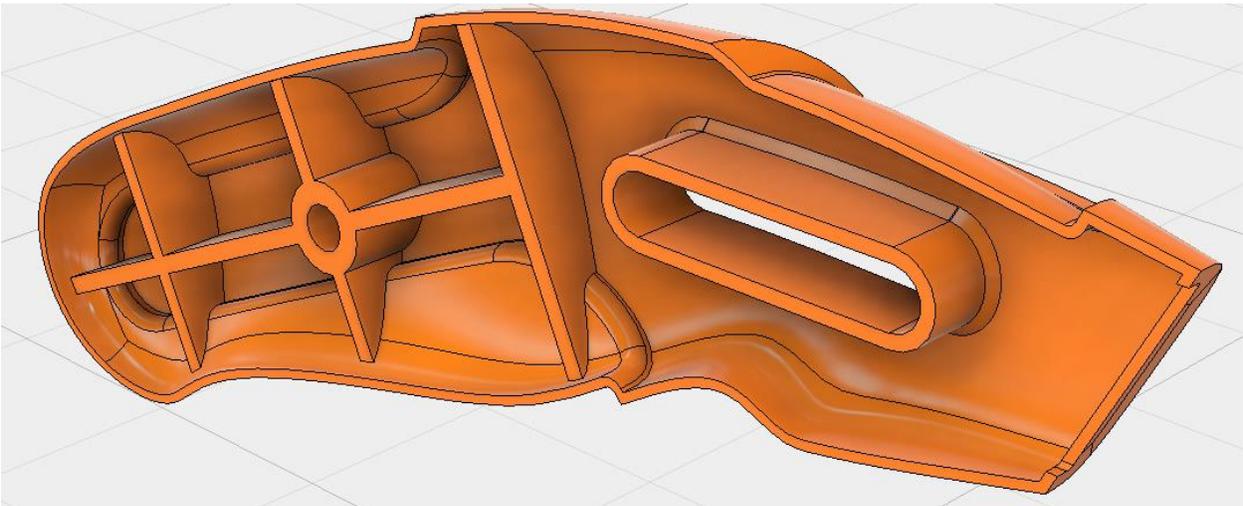
Use model to create designs with hard edges and flat faces. Model creates bodies requiring exact sizes and edges. Entering exact values is not required but is typical.



Use the sculpt workspace to create bodies with organic shapes. Sculpt bodies are highly curved and the shape is more critical than exact size.



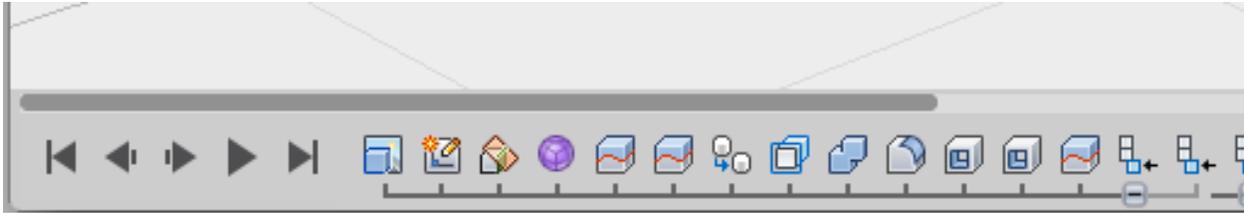
Very frequently, your designs will require that you work in both sculpt and model workspaces, back and forth. You might even throw patch in there to stitch surfaces together into a solid. You can work entirely in sculpt, entirely in model, or you can combine the two. You can combine sculpt and model to create the shape required as well as precise manufacturing features. Create the organic shape in sculpt then use model for manufacturing features afterwards.



Design History

Fusion 360 can work with or without recording design history. Design history refers to the operations you perform on the design to create and modify geometry. Operations are recorded in the timeline at the bottom of the interface.

Autodesk Fusion 360: Introduction



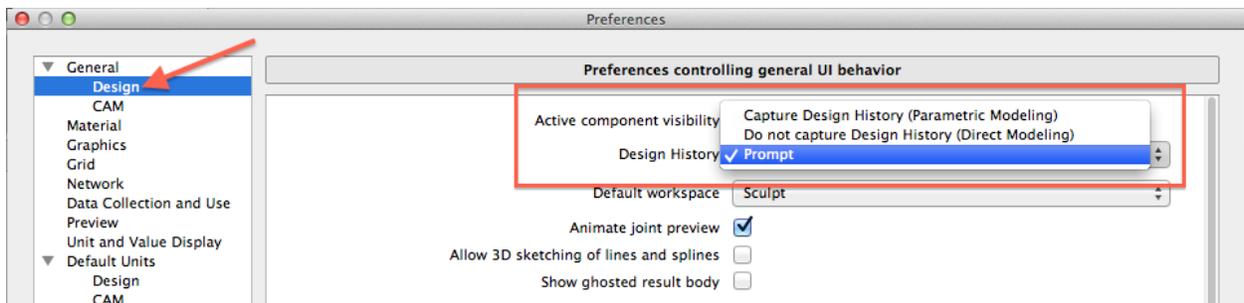
When using parametric modeling, the design history is captured in the timeline at the bottom of the interface. Operations are captured in the order they are performed in. History is captured for commands in the Model and Patch workspaces. You edit the operations in the timeline to make changes to your design.

When using direct modeling, design history is not captured. The same commands are used from the toolbar but there is no timeline. You use commands like Press Pull or Move to move faces and change your design.

So, why use one over the other? Using history allows you to make precise predictable edits to one or many components and allow the model to rebuild reliably. History is also useful if you plan to go switch between the model and sculpt workspaces. This allows you to create your outer shape, then create model operations (shell, split, hole, etc.) then go back and change your shape. If history is enabled, the model operations will recalculate to fit the new shape.

With direct modeling (history is off), you change geometry by moving faces. There are no operations to edit and therefore, no relationships between features in the design. Direct modeling works well for quick concept design or when working the imported data.

You can control the default behavior for new designs using preferences or you can turn design history on/off in the browser after a model is created.



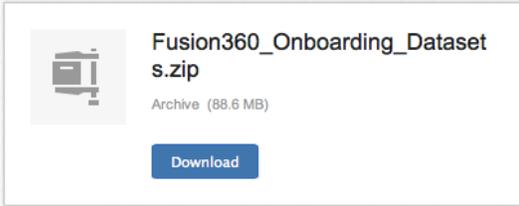
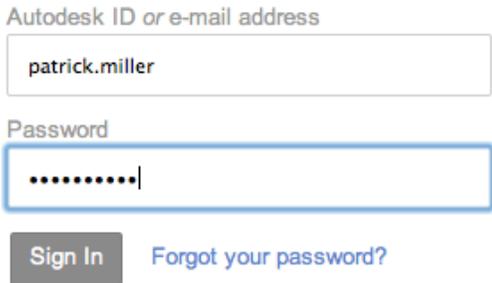
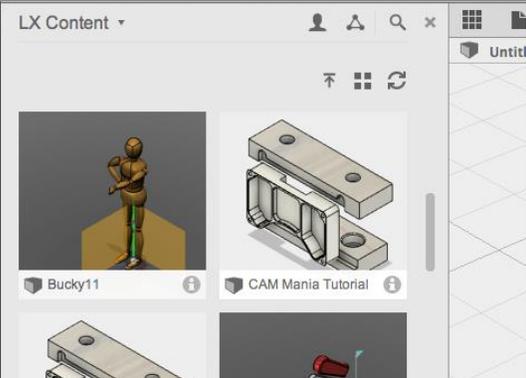
Autodesk A360

Fusion 360 uses Autodesk A360 to manage data and collaborate with teams. When you access your designs in Fusion, you are actually using A360 under the hood. You can also access your data in a web browser using A360. A360 provides access and management of the versions of your designs. You can

Autodesk Fusion 360: Introduction

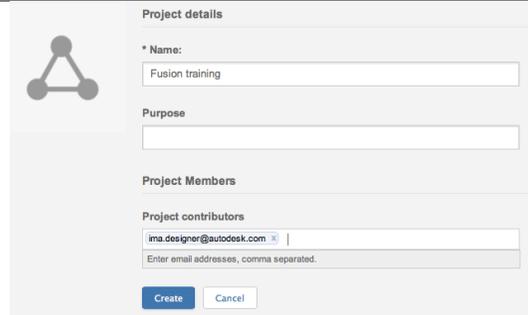
also upload other supporting design documentation. A360 lets you manage who can access your design data. All these tools within A360 make it easy to collaborate with team members.

Get Started: To get started with Fusion 360, we will log in, create a project, and save a design to the project.

<p>Step 1 – Download files</p> <ol style="list-style-type: none">1. Click the link below to access the zipped dataset. If clicking the link does not work, you can copy and paste the link into a browser. https://myhub.autodesk360.com/ue29c9141/shares/public/SH7f1edQT22b515c761e12e3a50ab9cda319 <p></p> <ol style="list-style-type: none">2. Click  to start the download.3. When the download is complete, unzip the files. Make note of where you put the files. You will use these throughout the training.	 <p>Launch Video</p>
<p>Step 2 – Log in to Fusion</p> <ol style="list-style-type: none">1. Launch Fusion 360.2. Use your Autodesk ID to log in to Fusion. <p>If you do not have an Autodesk ID, you can create an account for free.</p>	
<p>Step 3 – Open the Data Panel</p> <ol style="list-style-type: none">1. Click  to display the Data Panel.2. The data panel slides open on the left.	

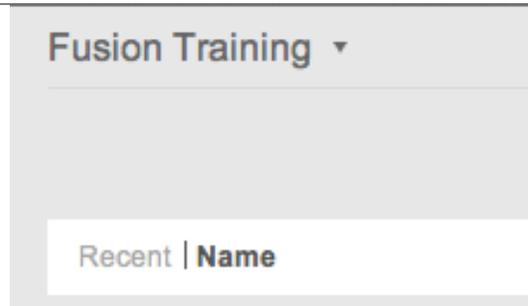
Step 4 – Create a new project

1. Click  to go to Autodesk A360 in your Internet browser.
2. Move the mouse over  then click  to begin creating a new project.
3. Enter **Fusion Training** in the Name field.
4. Enter email address in the Project Contributors field to invite people to this project.



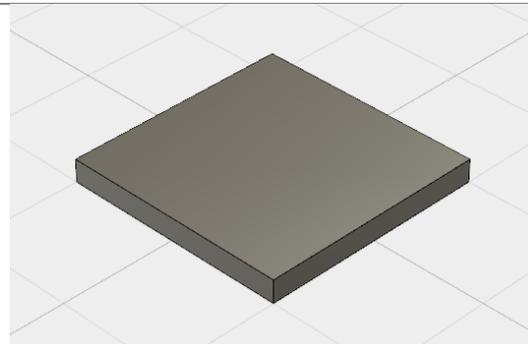
Step 5 – Create a new design

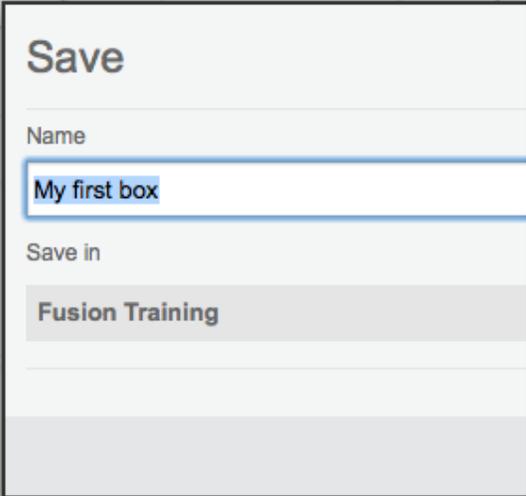
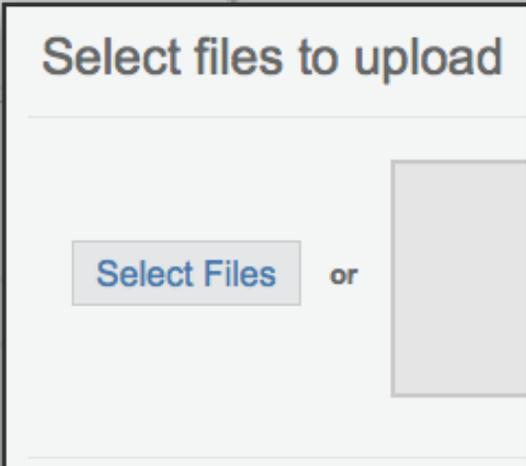
1. Return to Fusion.
2. Click  to force a refresh of the data panel.
3. Click the project name at the top of the data panel then select Fusion Training from the list. The data panel refreshes. There will be no designs listed because this is a new project.
4. Click  and select New Design. A new empty design is created.



Step 6 – Add geometry to the design

1. Click Create > **Box** to start the box command.
2. Select the plane along the bottom of the design (XZ plane).
3. Pick two points to define the length and width of the box.
4. In the Box dialog box use these values:
Length: **100 mm**
Width: **100 mm**
Height: **10 mm**
5. Click OK.



<p>Add notes or tips here, optional depending on complexity of step/workflow</p>	
<p>Step 7 – Save the design</p>  <ol style="list-style-type: none"> 1. Click  to save the design. 2. Enter My first box in the Name field. 3. Verify that Save in is set to Fusion Training. 4. Click Save. The design is uploaded and added to the Fusion Training project. A thumbnail is added to the data panel for this design. 	
<p>Step 8 – Upload designs</p>  <ol style="list-style-type: none"> 1. Click  to upload designs to the Fusion Training project. 2. Click Select Files. 3. Browse to the location where you have the training files and select the required designs. 4. Verify that Uploading to is set to Fusion Training. 5. Click Upload. <p>The Upload progress dialog displays when files are uploaded. You can close this dialog and continue working. The Upload progress is shown at the bottom of the data panel.</p>	

Hot Keys

Command	Windows	Mac
Undo	Ctrl + Z	Command + Z
Redo	Ctrl + Y	Command + Y

Autodesk Fusion 360: Introduction

Copy	Ctrl + C	Command + C
Paste	Ctrl + V	Command + V
Cut	Ctrl + X	Command + X

Sculpt Workspace Selection	Windows	Mac
Grow selection	Shift + Up arrow	Shift + Up arrow
Shrink selection	Shift + Down arrow	Shift + Down arrow
Loop selection	Alt + P	Control + P
Loop grow selection	Alt + O	Control + O
Ring selection	Alt + L	Control + L
Ring grow selection	Alt + K	Control + K
Ring shrink selection	Alt + J	Control + J
Previous U	Alt + Left arrow	Control + Command + Left arrow
Next U	Alt + Right arrow	Control + Command + Right arrow
Previous V	Alt + Down arrow	Control + Command + Down arrow
Next V	Alt + Up arrow	Control + Command + Up arrow
Range selection	Alt + M	Command + M
Invert selection	Alt + N	Command + N
Toggle box mode	Ctrl + 1	Ctrl + 1
Toggle control frame mode	Ctrl + 2	Ctrl + 2
Toggle smooth mode	Ctrl + 3	Ctrl + 3
Select edge ring	Double-click an edge	Double-click an edge
Select face ring	Select two faces then double-click a third face	Select two faces then double-click a third face

Edit Form Command	Windows	Mac
Add geometry	Alt + Drag	Option + Drag
Add geometry and keep creases	Alt + Ctrl + Drag	Option + Command + Drag