# Fusion 360 Academy

## Basics



## Going 3D



## Managing your work



## Advanced

# The Basics

**Welcome to the Robotics Society Fusion 360 Academy.**

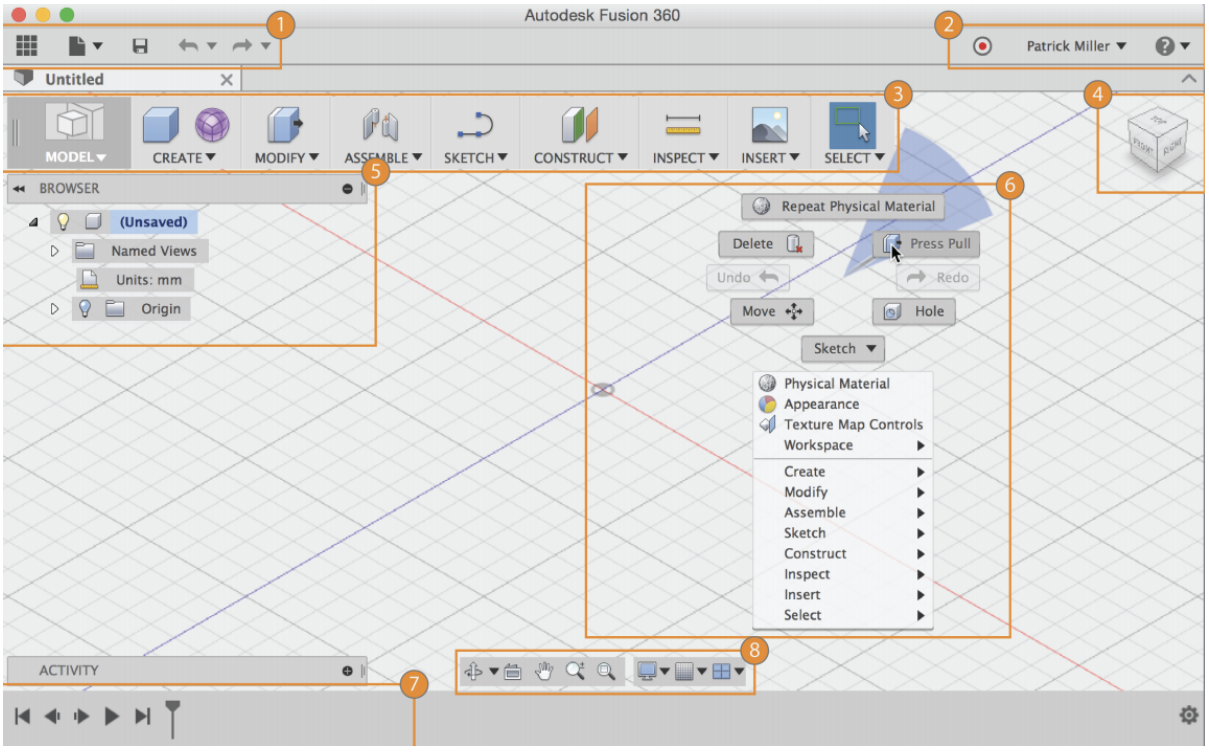
Don't worry if you're new to 3D design, we'll start right from the basics & cover all the terminology and essential steps.

The CAD software we'll use in this workshop is called **Fusion 360** but it's not too different to other packages you may have heard of like **solidworks** and **sketchup.**

If you haven't already, follow the other guide to get signed up on Fusion for free.

## UI Layout

Once you get logged in you'll see a screen like below;




Don't panic! It's not as confusing as it may seem. The 8 sections labelled above are;

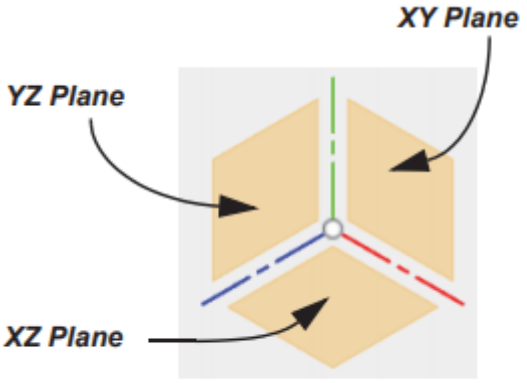
1. The **Application Bar** where you'll find the following essential buttons.

| | |
|---|---|
| | **Data Panel** – Used for data management and collaboration. |
| | **File** – Create a 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/redo operations. |

2. The **Profile section** where you can change user preferences like inverting scrolling & **improving performance** by choosing simplified animations (highly recommended)
3. The **Toolbar** where you'll select tools to use.
4. The **Viewcube** which tells you from which perspective you're viewing your model. Can be clicked and dragged to change perspective
5. The **Browser** where you can navigate through your objects & sketches
6. The **Marking menu** which provides shortcuts when you right-click
7. The **Timeline** where you can explore the development of your project and delete/modify steps from the past (covered later)
8. The **Navigation bar** which has interaction buttons like pan, grab & zoom

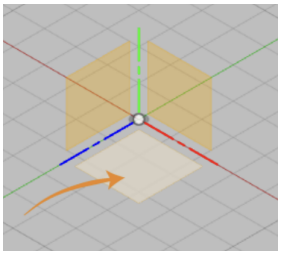
**Planes (not the flying things)**

Planes are the flat surfaces that when combined make up 3D space.



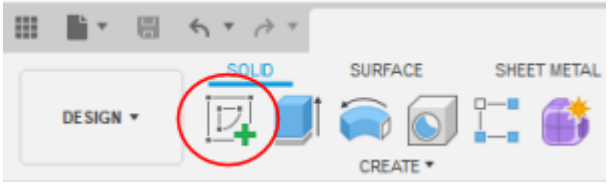

You'll be familiar with **X & Y** which together are used to represent 2D objects.

When you add the third plane **Z** you are then able to define 3D objects.

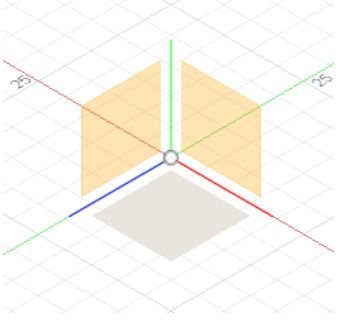


In Fusion X is **red,** Y is **green** & Z is **blue.** Which plane you think of as "Ground" doesn't really matter but we typically pick **XZ.** Don't worry about this too much yet, it'll become clearer when we get into modelling.

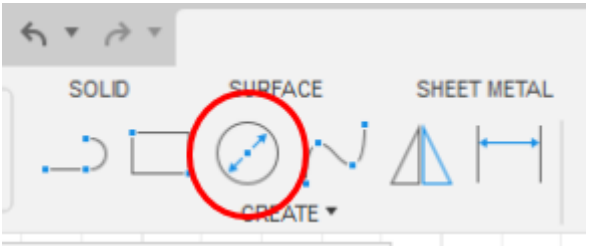
**Sketching**




1. To get started with sketching you firstly need to click **"Create Sketch"** in the top left create section**.**
2. You'll now notice the **planes** appear highlighted. Left click on the "Ground" (XZ) plane (Bottom-Middle one).



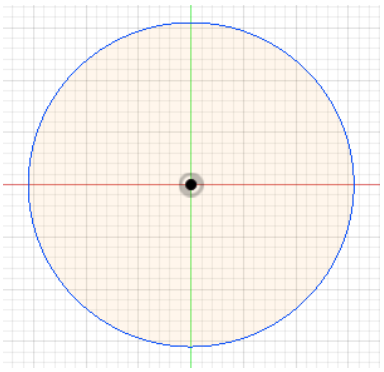
3. We're now viewing the XZ plane in sketching mode. This means we can now draw a shape. Here's how you'd draw a circle;



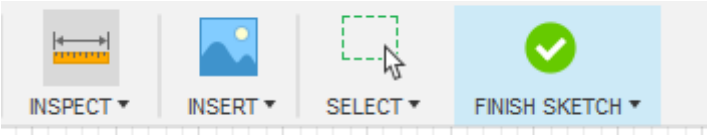

   a. Firstly select **Center Diameter Circle** in the create section**.** (Or press **c** as a shortcut)

   b. Next pick your **origin point**. The centre point is a good spot.

   c. Move the mouse away from the center to any other point to define the size, then left click to finish the circle.



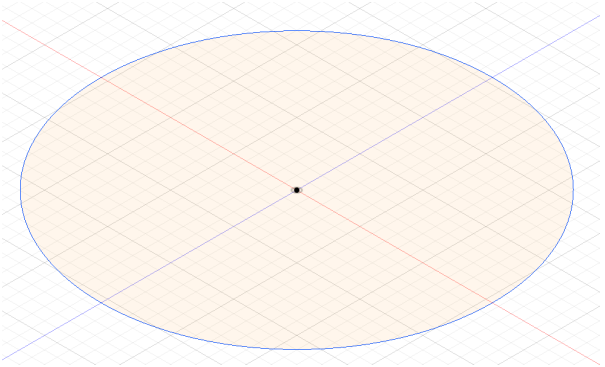
4. Now our shape is complete. Let's exit sketch mode by pressing **finish sketch** in the top right**.**




   If you click **home** next to the **Viewcube** (to return to the default orientation) you should see your shape like below.


The process for adding any other shape is the same and you can add multiple shapes during the same sketch.

Some common shapes include **rectangle**, **arc** or **line,** all of which entering sketch mode, picking a start & end point, and exiting when done**.**



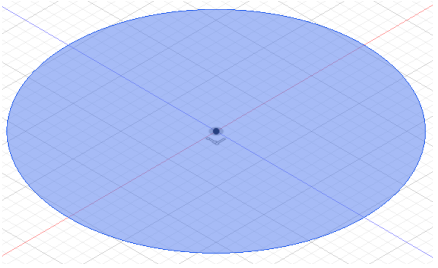
# Going 3D

**Take some time to get familiar with drawing 2D shapes before starting this section.** Going 3D only extends what we've done so far by allowing us to create & cut objects based on the sketches we make.

## Extruding

The first and most common way to make a sketch into a 3D object is extruding.

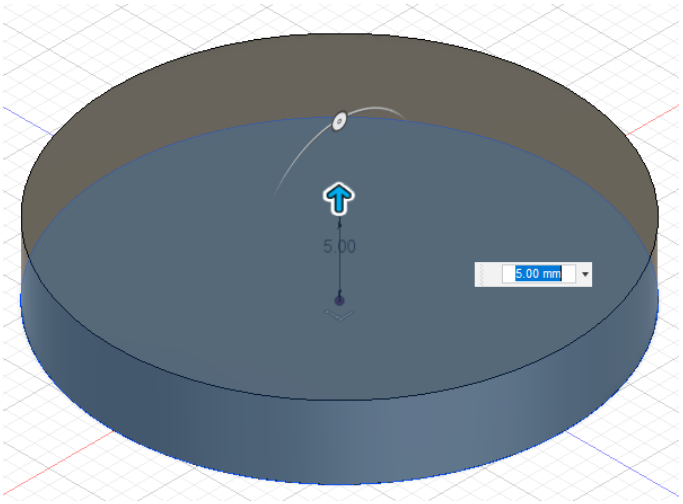
1. Start by ensuring your in select mode (i.e. not currently drawing something). You can tell this as when you hover over a sketch it will highlight blue

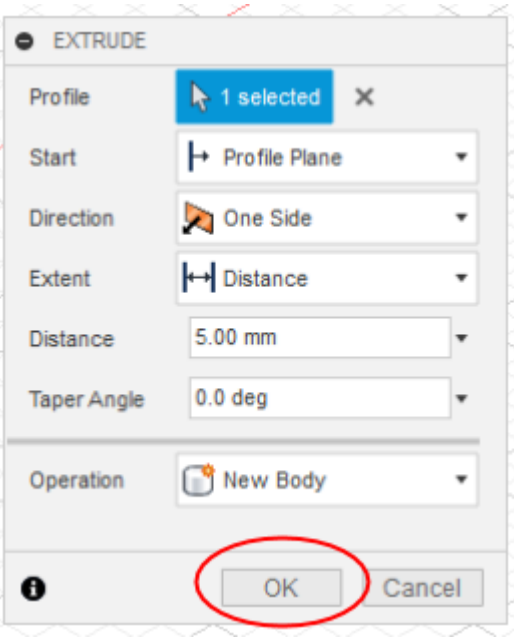


2. Select this sketch by **left-clicking it** (tip: you can select multiple sketches by holding **shift**)

3. Select **extrude** from the create section (Or press **e** as a shortcut)


4. **Drag the arrow** or type the height you want into the text box (pictured left). When you're happy select OK in the extrude dialog (pictured right)

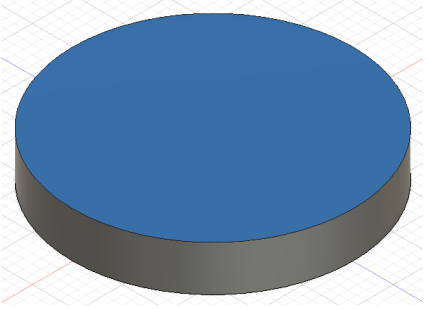


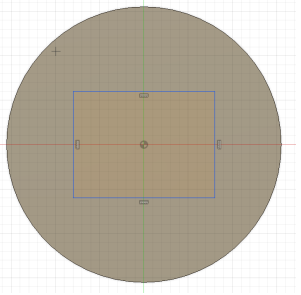

**You now have a 3D object!**

## Cutting

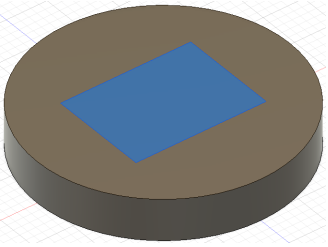
We can also use sketches to cut sections out of 3D objects.



1. Start by creating a sketch that aligns with the object you want to cut out. The easiest way to do this is to **left-click** a flat surface on a 3D object and click **Create Sketch** in the design section



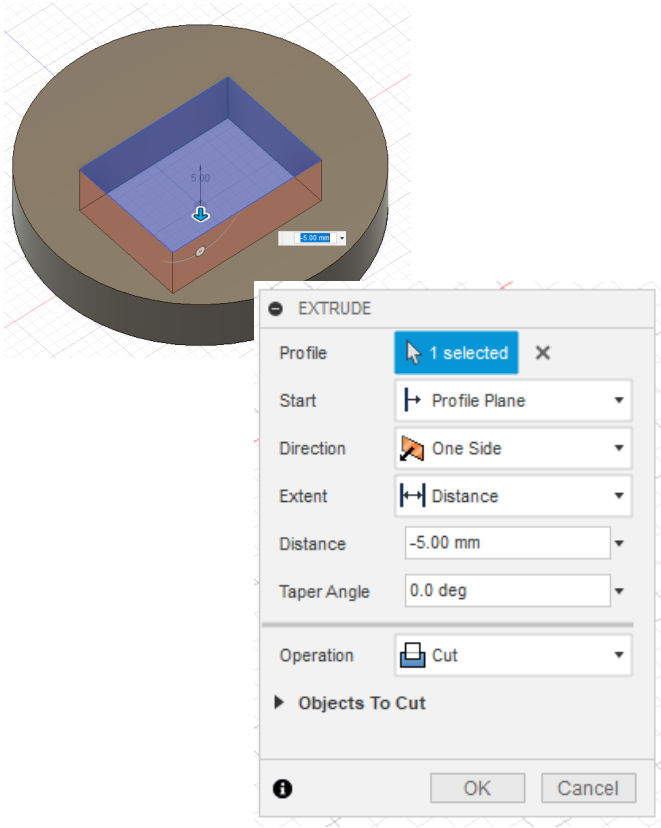
2. Next draw the shape you want to cut out. As an example I've created a rectangle.

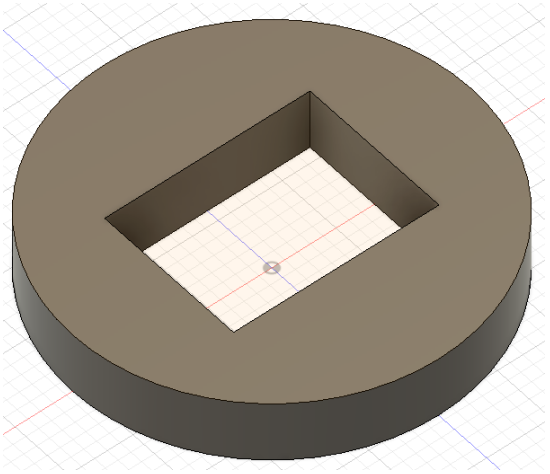


3. Press **Finish sketch** or **select** (both in the top write) to get out of drawing mode

4. Select the sketch you just created and select **extrude** (or press **e**)

5. **Drag the arrow into the body** (or type a negative distance into the text input). Notice here how the operation now says **cut** instead of extrude. When you're happy press OK.

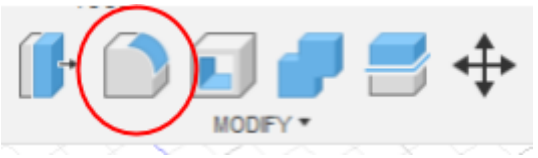


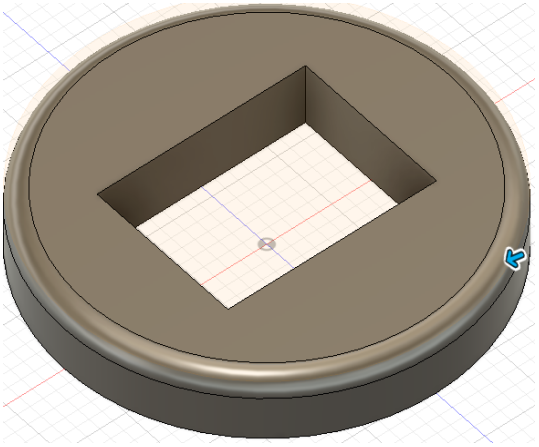
## Filleting

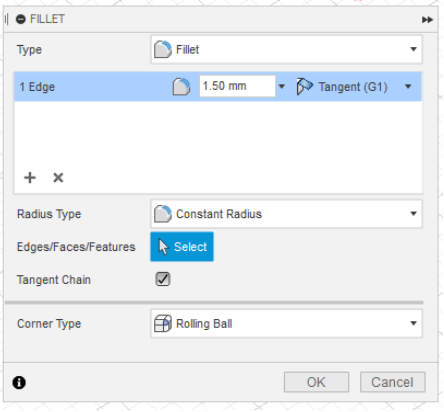
Another commonly used tool in on 3D objects is Filleting. This allows you to smooth edges. You can also use the **chamfer** tool the same way but for a straight edge.

1. Select the edge of the object you wish to fillet
2. Select **fillet** from the modify section (or press **f**)




3. **Drag the arrow** or enter a value for how much you want to smooth. When you're happy press OK.





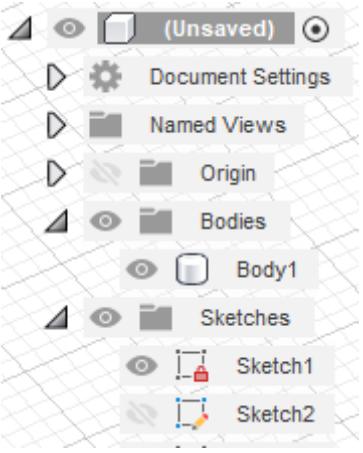

## Navigation

As you start creating 3D objects it's important to be able to view and manage them effectively.

All the important tools for navigating 3D space are found at the bottom.

- **Orbit (Hold shift & middle-mouse)**. Allows you to rotate your view
- **Pan (Hold middle-mouse)**. Allows you to move up, down, left & right based on your current view angle

Also make good use of the **browser** on the left to keep track of your **bodies & sketches.** These can be **renamed, hidden & edited** from this menu

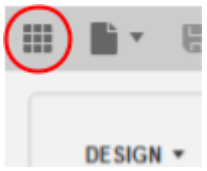


# Managing your work

Once you're starting to get comfortable with your work, you 'll want to organise designs better. Let's look at how to do this

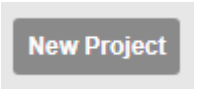
First, it is important to note that Fusion 360 file storage is completely **cloud based**. This means you can manage and view your files online and can log into Fusion 360 on **any computer** and have all of your work available.

## Managing files & sharing



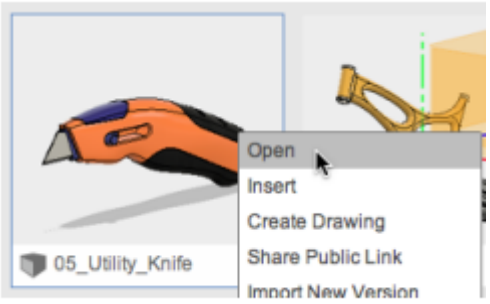

All file related actions can be found in the **data panel** which is the icon in the top left of the screen.

Here you'll notice the **New Project** button which creates a project folder for you to keep your files in.




You will also find all your created designs which can be moved into projects.

If you **right click** a design you will also see a variety of options including **Share Public Link.** This lets you share a link to your design that even someone without Fusion 360 installed will be able to view your model and look at the list of parts etc.




## Collaborating with Others

At the top of the project data panel there are two buttons, one for **data** (the models and sketches that make up your project) and one for **people** (the people involved in the project).


Because Fusion 360 stores all the files in the cloud it is easy to invite other people to collaborate on a project via the people tab. This will give them access to view and edit the project files so multiple people in a team can seamlessly collaborate on a project together.

## Managing your timeline

When your designs get more complicated it's important to approach them in a way where you can easily modify components after they're made.

At the bottom of your screen you'll find the timeline. You can drag the **current time** (circled red below) to see the steps you went through to produce the final result. You can also **double-click** any step in the process to modify it.

Try to reduce your steps by;

- Create multiple shapes during a single sketch
- Using tools like pattern & mirror to reduce duplicated actions
- Apply actions to objects at the same time where applicable

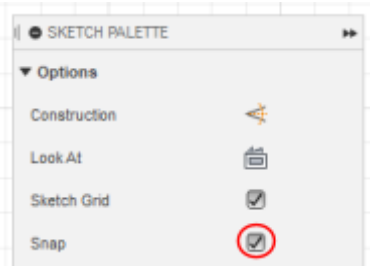
This will make you work far easier to maintain and update.

## Advanced

### Sketching

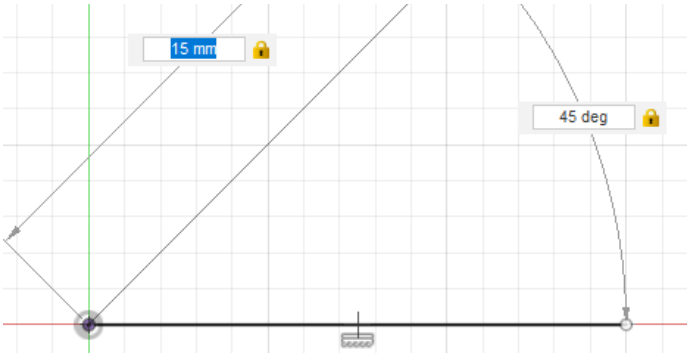
Dimensioning

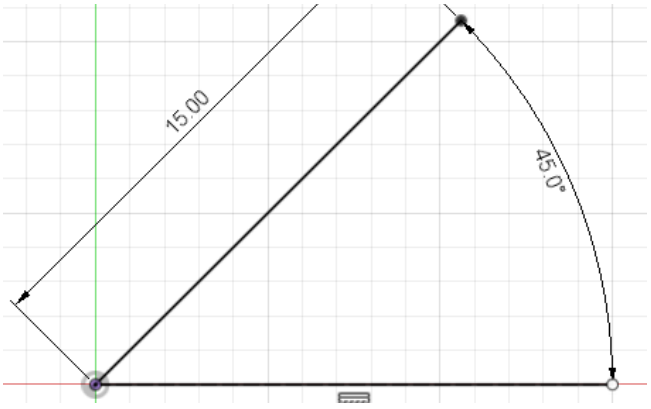
To improve accuracy whilst sketching firstly enable **snap** which will snap your mouse to key grid points.




Also always enforce the **length and angle** by entering these values. Once you do a lock symbol will appear next to the values.

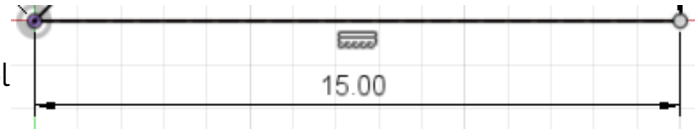
When you confirm these dimensions & hit enter they will be labelled.




These values can be modified by **double-clicking** them. You can also set dimensions after a sketch is created.

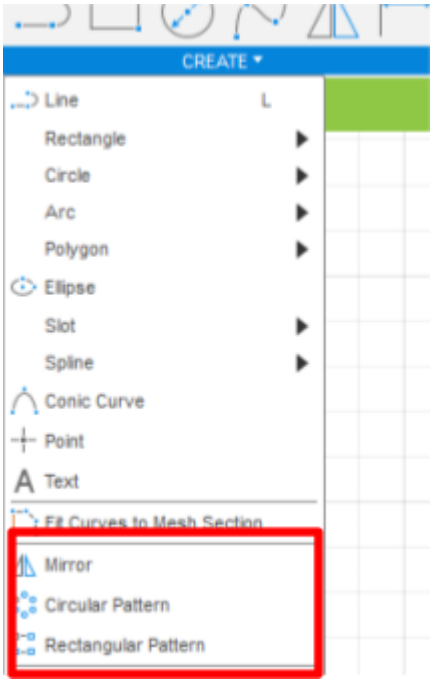

The **sketch dimension tool** can be found in the **create tab** on the top toolbar or by pressing the **d** key. With the dimension tool



selected you can click a line to set its dimension, or click 1 line and then another to set the distance or angle between them.

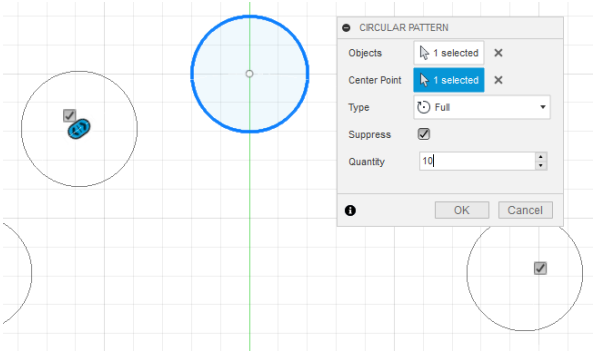
Patterns

To repeat sketches reliably you should make use of **mirrors & patterns**. The available options can be found under at the bottom of the create menu.

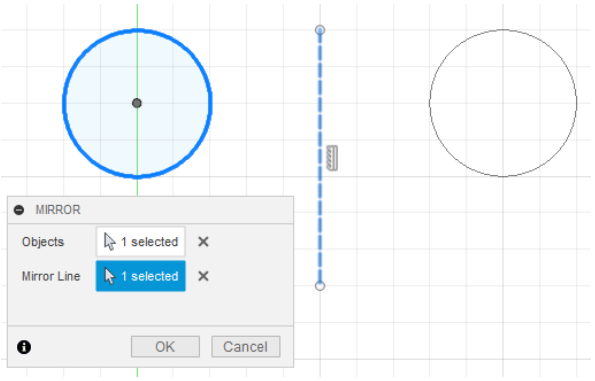



To create a circular pattern select the sketch you want repeated, the center point and how many times you want it copied.

Rectangular patterns are similar but allow you to set directions.

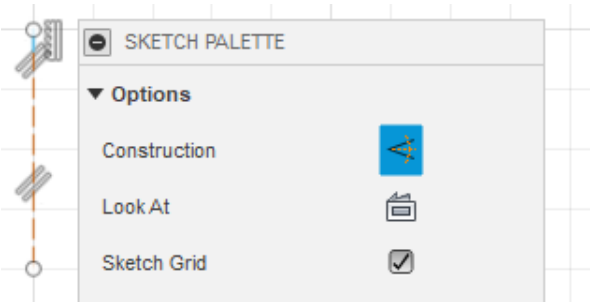



Mirrors work similarly but require a mirror line.




Construction lines

Another small but useful tool in sketching is **construction lines**. These are useful to create lines that shouldn't appear in the final sketch, but help with creating it.



To make a construction line simply press the **construction** box in the sketch palette and draw your shape.

Common uses for these lines include labelling distances/angles in sketches, marking midpoints, mirror lines and much more.

Constraints

Constraints are very useful for keeping your sketch clean and simple but also ensuring it represents what you intend.


The constraints tools can be found in the top toolbar. All of the constraints are useful, but the most commonly used are:

- **Tangent** - ensure a line and a circle meet at a single point.
- **Equal** - applied to two lines makes them equal length
- **Parallel** - applied to 2 lines makes them parallel
- **Perpendicular** - applied to 2 lines makes them perpendicular
- **Concentric** - applied to two circles (or arcs) makes the circles share a centre point
- **Collinear** - forces two lines to share a single axis, i.e. they both lie on the same infinitely long line

**For example**, when constructing a square you can define all the edges as equal length and two of the lines perpendicular. Then you can use the dimension tool to set the length of the side and it can be any size desired and rotated relative to anything else but it will always be a square.

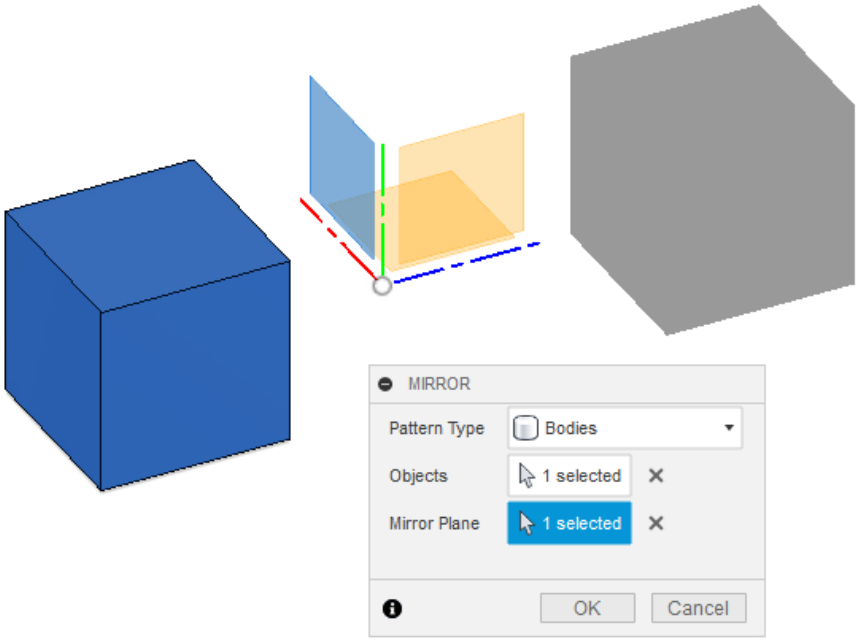
Many constraints are applied to the sketch automatically by Fusion 360, such as the **horizontal/vertical** constraint (when drawing a line that snaps to the grid lines), and **coincident** constraint (when starting a new line from the end of another the points that define the ends of the line are coincident).

Constraints applied to the sketch can be seen on the sketch as **greyed out versions** of the symbols in the toolbar. Clicking on these symbols allows you to select, then **delete** them if you wish.

**Modelling**

Pattern & Mirroring

The same tools for reproducing entities are supported in objects as well as sketching.

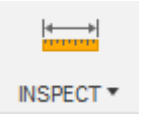



Using the tools works fairly similarly with the main difference being how you select objects.

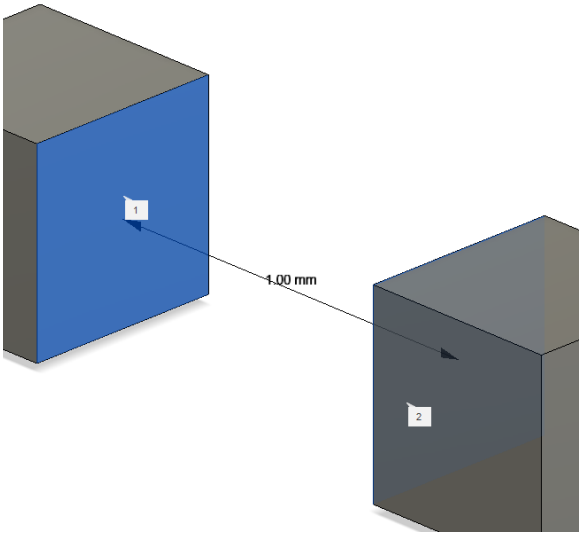
You will almost always want to copy object **bodies** as opposed to faces so make sure you've selected this first. Then in the case of mirroring select the **mirror plane** and you should see a ghost preview object before clicking OK.

Measure

When working on a design you will also want to measure distances and angles. Fusion has a tool for this called **inspect**.




After selecting it you can click any two lines or faces and it will label distances between them and angles where applicable.

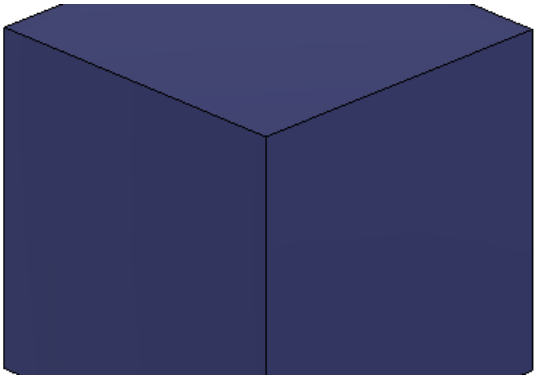


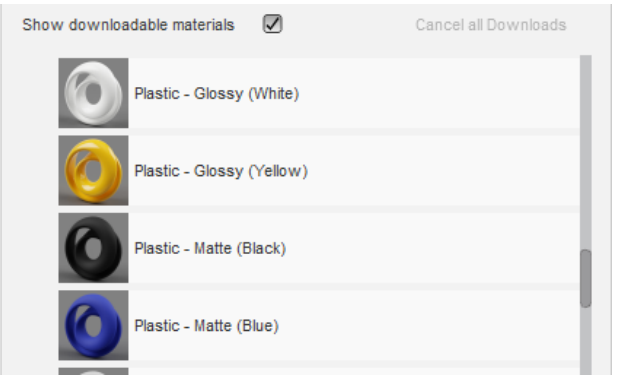

Appearances

Distinguishing objects in complex projects can be improved by colouring them, either as realistic representations (materials) or just highly contrasting colours.

To change the appearance **right-click** after selecting an object and press **Appearance** (or press **a**)

Once you do so the appearance menu will appear with a selection of materials in folders. To apply a colour simply **drag & drop** it from the menu onto the object






Add-ins

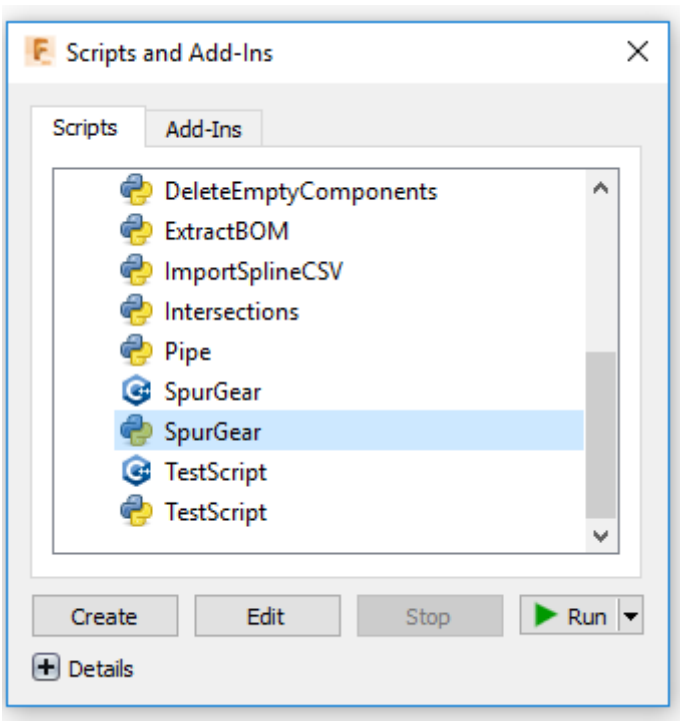
When attempting complex tasks that might have been done before, first consider checking out the Add-in section

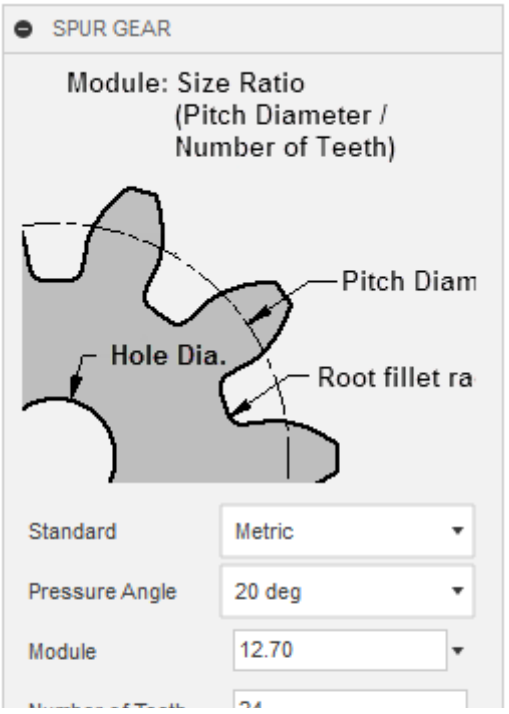
To find an Add-in go to the **tools** tab on the top menu.


From here you can see a selection of sample scripts that can be used to simplify complex object creation.

To use one simply select it (i.e. **SpurGear**) and press **Run.**

When you run a script you'll get a dialog to enter the required information and in the case of SpurGear this will create a new component.



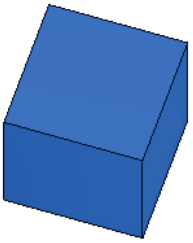

## Assembly

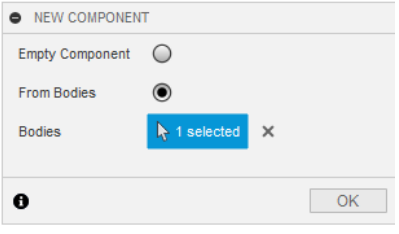
Another important element of 3D design is establishing how objects interact with one another. By doing this correctly we can easily move around and update objects in our designs.

**Components** are individually defined parts of a wider design. Making an object or group of objects a component gives us some additional abilities that we'll cover later.

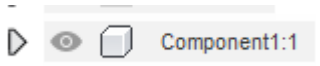
To create a component click the **New Component** button in the assemble section.


After this select **from bodies** and select which bodies you want to go into the component. Then press OK.



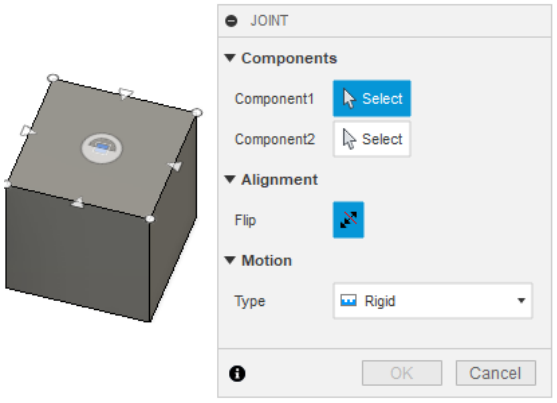


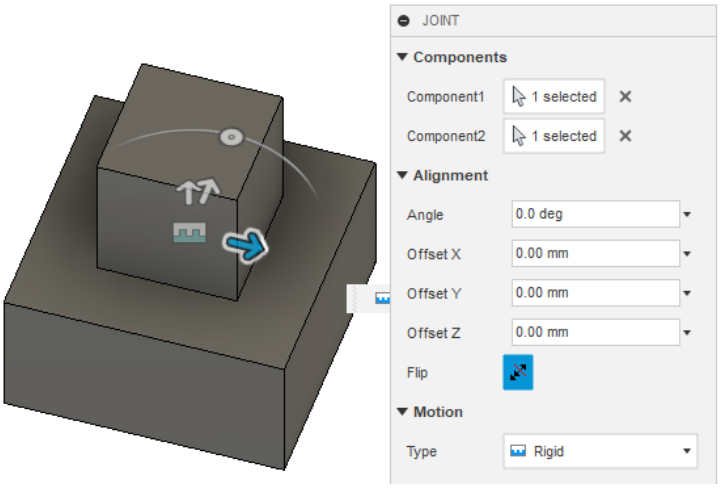
You'll notice the component appears in the **browser** list.




Once you've created multiple components you can create a joint. To create a joint click the button next to create component (or press **j**)

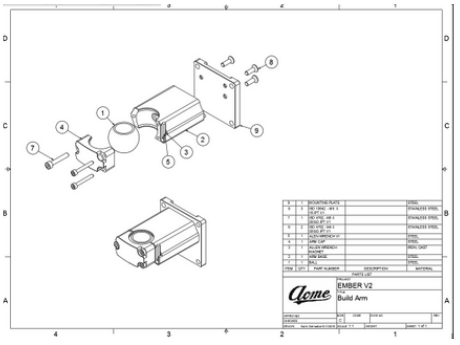

From here you can pick any two points from two components and when you do so the components will lock together. This is known as a **rigid** joint where the two components are locked together.
Once you press OK these components are now joined and so even if a component is updated, the joint will remain intact.




## Drawing & rendering

There are a variety of additional tools we won't cover in this guide. Two of the most useful are drawing & rendering.



**Drawing** allows you to take your design & make it 2D. Doing this you can export your design as a pdf or dwg which you can use for **laser cutting** or just to document your design.

**Rendering** allows you to create photo realistic images of your designs with set lighting & appearances.