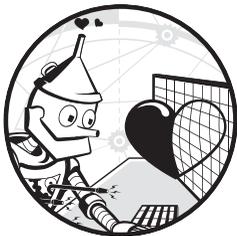# 3

DESIGNING YOUR FIRST MODEL

It's finally time to get modeling! Over the next few chapters, you'll gradually learn increasingly complex sketching tools, features, and techniques. By the end of the book, you'll be a master of 3D modeling and have the skills to design whatever you can think of. First, let's start with the basics.

Getting to Know Your Tools

In this chapter, you'll learn to use fundamental sketch tools and features in Fusion 360, which you'll use more than any of the complex tools in the chapters to come. Just like how a skilled carpenter can do amazing things with a handful of basic tools, you'll learn how to construct models using use simple tools like Extrude and Revolve. These foundational tools may not be glamorous or designed for specialized work, but they're powerful and useful for a wide variety of tasks.

Using Basic Sketch Tools

You can find the basic tools under the Sketch drop-down menu, as shown in Figure 3-1. If you don't see the Sketch drop-down menu, make sure you're in the Model workspace.
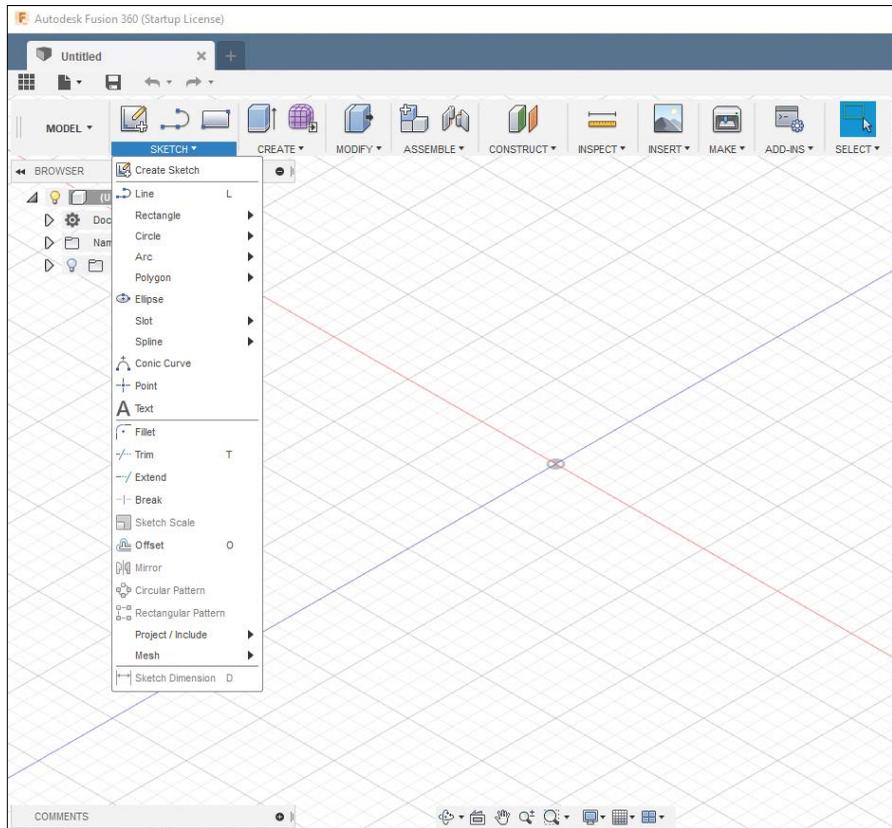

Figure 3-1: The Sketch tools menu

You should see quite a few options here, and many of them have additional submenus. In this chapter, you'll use the following tools:

Create Sketch   Creates a new sketch. When using this tool, you must select on which plane the sketch will be drawn. When you haven't yet started a model, you will usually select one of the x-y, y-z, or z-x planes, and if you've already started building a model, you'll usually select a flat face as a sketch plane. You can also start a sketch by selecting one of the sketch tools, like the Line tool, and then choosing a plane.

Line   Draws straight line segments. You can connect multiple segments together to form shapes. This tool is so frequently used it gets a default keyboard shortcut key: L.

Rectangle   Creates four line segments automatically joined with *constraints*, or rules that specify how sketch entities are linked. For

example, a constraint might lock the endpoints of two line segments together, make an arc tangent to a straight line, or make two lines parallel. In this case, the Rectangle tool automatically constrains the endpoints of the four lines and makes them perpendicular to each other. The submenu contains options for defining the rectangle, and the keyboard shortcut R draws two-point rectangles that are defined by any two corners.

Circle   Draws circles. Multiple options are available under the submenu for defining and positioning the circle. Sketch circles are true mathematical circles—they're not divided into arcs or a series of line segments. The C shortcut creates a circle defined by its centerpoint and diameter.

Arc   Draws arcs. The submenu provides a few options here for how you want to define the constraints of the arc. Since arcs aren't as common as the other shapes, the Arc tool doesn't have a keyboard shortcut.

Fillet   A *fillet* (pronounced "fill-it") is a frequently-used tool for rounding a sketch's corner. You define it by the radius of the arc that joins the two lines. When you use it in a sketch, this tool automatically cuts two line segments short, places an arc connecting them, and constrains that arc to be tangent to both line segments.

Trim   When two lines intersect, this tool trims one of the lines down to the point where the intersection occurs. The keyboard shortcut is T.

Extend   Extends a line until the point where it intersects with another line. If you apply this tool and there are no other lines to intersect, then nothing will happen—lines can't go on infinitely in Fusion 360.

Offset   Creates a line, or chain of lines, that is offset from the selected line by a defined distance. The keyboard shortcut is O.

Sketch Dimension   Defines the dimensions of your sketches, like the length of a line or the angle between two lines. The keyboard shortcut is D.

The sketch entities you create with these tools will likely form the bulk of your toolset—you can sketch almost anything with just these tools. But you're not reading this book just to make sketches—you want to create 3D models! That's where features come in.

Using Initial Features

You can use the tools in the Create menu (shown in Figure 3-2) to make initial features, which you'll combine to create 3D shapes. You must always use one of these initial features to create your first 3D entity because they don't require any existing model geometry to work off of (though a sketch may be required). This contrasts with the tools from the Modify menu, which only work with existing geometry. Keep in mind that you can use these tools to create features throughout the modeling process, and not just to create initial features.
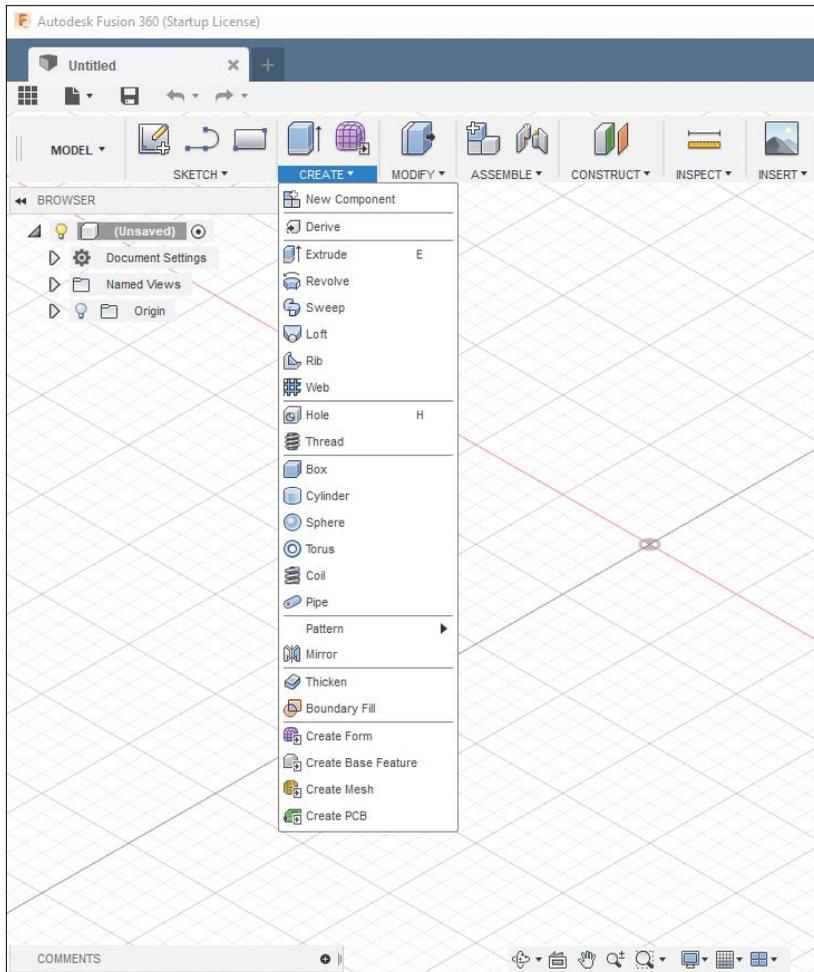
Figure 3-2: Initial features are located under the Create drop-down menu.

Some of these initial features are more frequently used than others. The following list briefly describes the most common features:

Extrude   Adds thickness to a sketch to form a 3D solid. Conversely, an extrude feature is also used to cut into an existing solid in the shape of a selected sketch. This is the most common feature in 3D modeling, and the keyboard shortcut is E.

Revolve   Similar to Extrude, this feature creates a new solid or cuts an existing solid from a sketch. The Revolve feature makes a solid body by revolving the sketch around a selected center axis by a specified number of degrees. For example, a square that touches the axis of revolution would form a cylinder if it were revolved a full 360 degrees.

Hole   Streamlines the process of cutting a hole in a solid. With Extrude and Revolve, you can create a hole of any shape you can sketch, but using

the Hole feature can sometimes be faster for creating simple holes. But since cutting holes with Extrude or Revolve is more common and versatile, I typically avoid using Hole.

Box, Cylinder, Sphere, Torus, Coil, and Pipe   These features create primitive solids. Like the Hole feature, these features can all save time in some situations—mostly as basic starting points—but for learning purposes, you won't be using them in this book. You can create each of these (with the possible exception of the Coil feature) with Extrude or Revolve.

You might be surprised that you'll only use two of these initial features for now (Extrude and Revolve), but they're powerful and useful. In fact, Extrude and Revolve alone will probably make up more than 90 percent of the initial features you use in your 3D modeling endeavors.

Modifying Features

While you can create initial features without any existing solids, you'll need to have a 3D object to modify features. Modifying tools are often used toward the end of the modeling process to put the finishing touches on a part you've designed, but you can certainly use them earlier. You can access these tools from the Modify drop-down menu (see Figure 3-3).
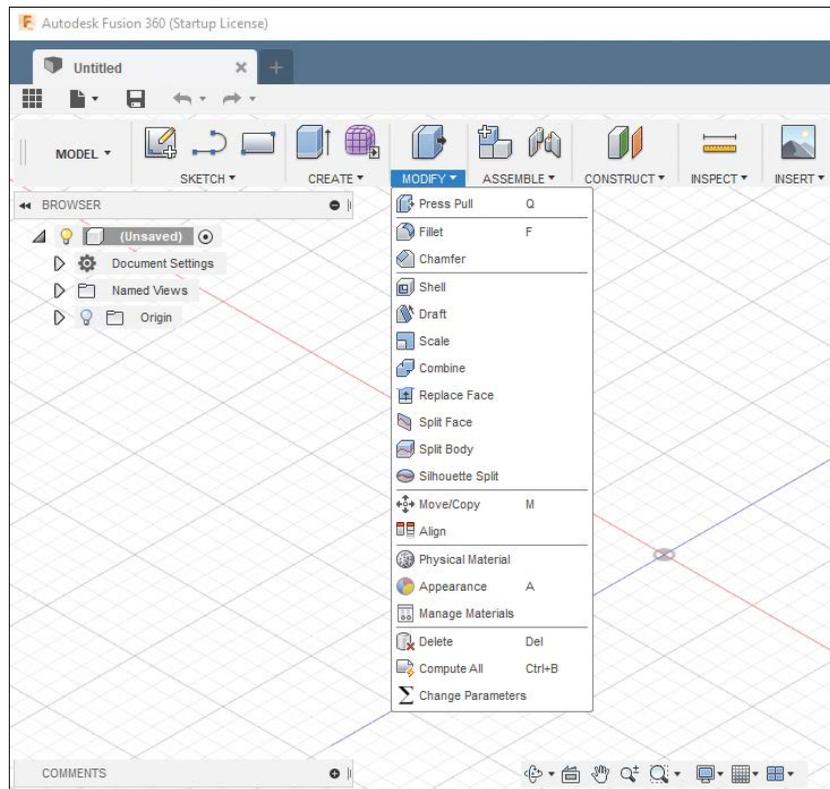

Figure 3-3: You can find modifying features under the Modify drop-down menu.

Like the initial features, you'll use some of these tools more than others, so the following list covers just a few of the available modifying tools in Autodesk Fusion 360:

Fillet   Like the Fillet sketch tool, you can use the Fillet modifying tool to create rounded edges and corners three-dimensionally. You can select inside or outside edges and define a radius. The keyboard shortcut is F.

Chamfer   The Chamfer tool works much in the same way as Fillet, except that it gives the edge a bevel instead of rounding it. You can define a chamfer by the angle of the cut and one distance from an edge, or with the distances from two edges.

Shell   A surprisingly useful feature, the Shell tool hollows out a solid and removes the selected face for an opening. You also need to specify the thickness of the walls, so the model remains a solid. Shell could, for instance, turn a cylinder into a cup or a cube into a box.

Now that you know what's in your tool belt, let's jump into the modeling!

Modeling a Cube

Creating a cube is like the "Hello, World!" of 3D modeling. Although you can quickly and easily create a cube with the Box initial feature, you'll create a cube the hard way from sketches so you can learn about the modeling process.

Creating the Sketch

To model the cube using an Extrude feature, you first need to start with a sketch. Go to the **Sketch** drop-down menu and select the **Create Sketch** option. You should be asked to choose a plane on which to draw the sketch. Figure 3-4 shows the sketch on the xy-plane. The plane you choose is completely up to you—you can always rotate models at a later point, so it's really about how you want to picture the part in your mind.

Once you choose your sketch plane, the viewport should rotate to face that plane. You can rotate the view if you like, but most of the time you'll want to sketch while facing the sketch plane.

Because you're modeling a cube, start by sketching a rectangle. In this case, you want a *center rectangle*. This means the rectangle will be constrained to stay centered on the origin point, or (0, 0, 0). Constraints are geometric rules that apply to sketch entities. For instance, a perpendicular constraint forces two lines to meet at a 90 degree angle. Keeping your models constrained to the origin is typically good practice, because it will make using the default reference planes and axes easier in the future.

Now navigate to **Sketch ▸ Rectangle ▸ Center Rectangle**. A dialog offering various options should pop up on your screen. Clicking the plane determines the centerpoint, so click the origin to automatically create the rectangle's first constraint. Your next click should place a corner of the rectangle, so just click off in space a bit away from the origin so you get something rectangular, as shown in Figure 3-5. The point you click should be mirrored across both the x- and y-axes—automatically placing all four points of the rectangle.
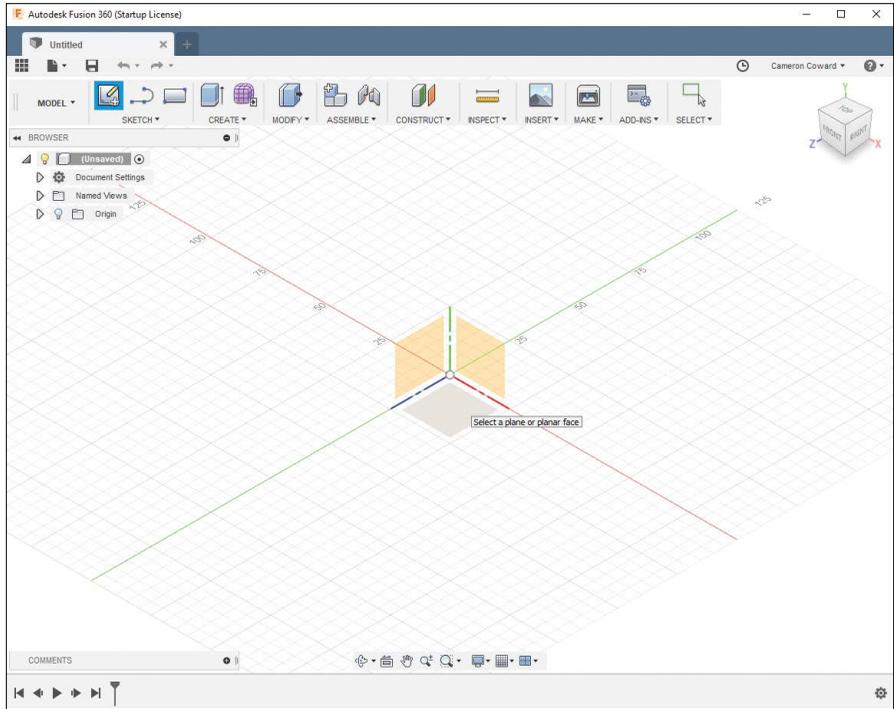
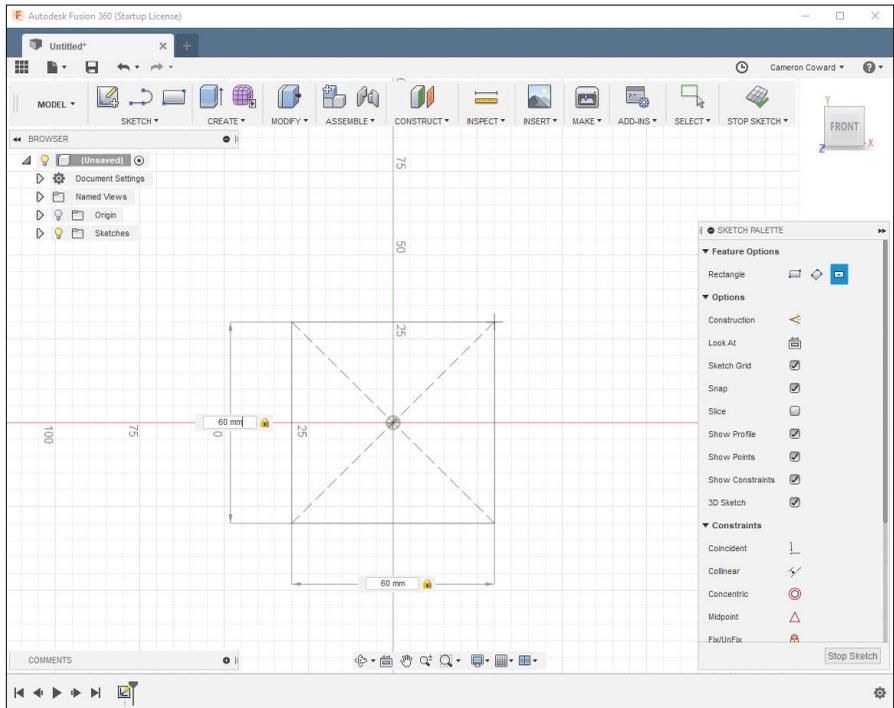Figure 3-4: Sketch on whichever plane makes the most sense to you.

Figure 3-5: When sketching, you can define dimensions during or after the operation.

You may have noticed that, after your second click that set the corner of the rectangle, the dimensions popped up for the width and height of the rectangle. You could have specified one of them, pressed TAB, entered dimensions for the other, and then pressed ENTER. This would have defined the dimensions of the rectangle and constrained it to them.

But because you didn't do that and just clicked a point in space, Fusion drew the rectangle without any associated dimensions. This means that the rectangle is constrained to be centered on the origin, the endpoints of the four lines are automatically locked together with coincident constraints, and the four lines are constrained to be perpendicular to each other. But, it doesn't have any dimensions to constrain its actual size.

Completely Constraining the Sketch

At this point, the four lines of your rectangle should be blue, which by default signifies that a sketch isn't completely constrained. You should never finish a sketch without it being completely constrained with geometrical constraints or dimensions. Until the sketch is completely constrained, points and lines could unintentionally shift, which leads to ambiguity about the accuracy of the part; and when it comes to CAD, ambiguity is always bad.

To completely constrain the rectangle, you need to define the dimensions of the part. Use the D shortcut to create a sketch dimension—or choose it from the Sketch menu—and select a vertical line. Move your cursor off to the side of the line and click to place the dimension. Then enter **50** and press ENTER. Do the same for one of the horizontal lines. The dimensions should display as shown in Figure 3-6.
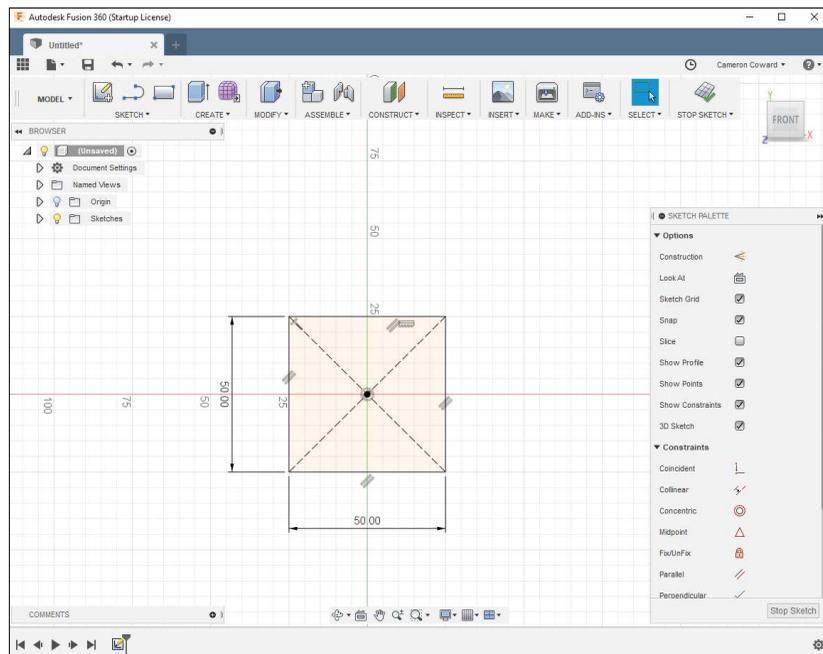

Figure 3-6: Sketch lines are black once they're fully constrained.

Now that you've defined the dimensions, all the lines should turn black in the default Photobooth visual environment (colors may look different if you change the environment). That means you've completely constrained the rectangle, and nothing unexpected should happen later on that might affect the precision of the sketch. Ensuring sketches are always fully constrained should keep you from having to chase down problems when you're working with more complex models. Leaving a sketch partially unconstrained won't break anything, but it can cause unpredictable results if the sketch or features shift.

Extruding the Sketch

To extrude your completely constrained sketch to form a 3D solid, press E or choose **Create ▸ Extrude**. The view should shift to make it easier to tell what your extrusion is doing, and you'll be asked to choose a *profile*, which is any closed loop in a sketch. In this case, your profile is your rectangle, so click somewhere inside it. Like in Figure 3-7, the selected profile should be highlighted. Although it doesn't apply here, you can select multiple profiles if they're on the same plane by clicking inside them when the "profile" field in the dialog is active.

Like all features, Extrude provides you with some options to work with. The Start option is where you want the *zero point*, or the start of the extrusion, to be. In this case, you should stick with the default, which is to start it at the sketch plane.
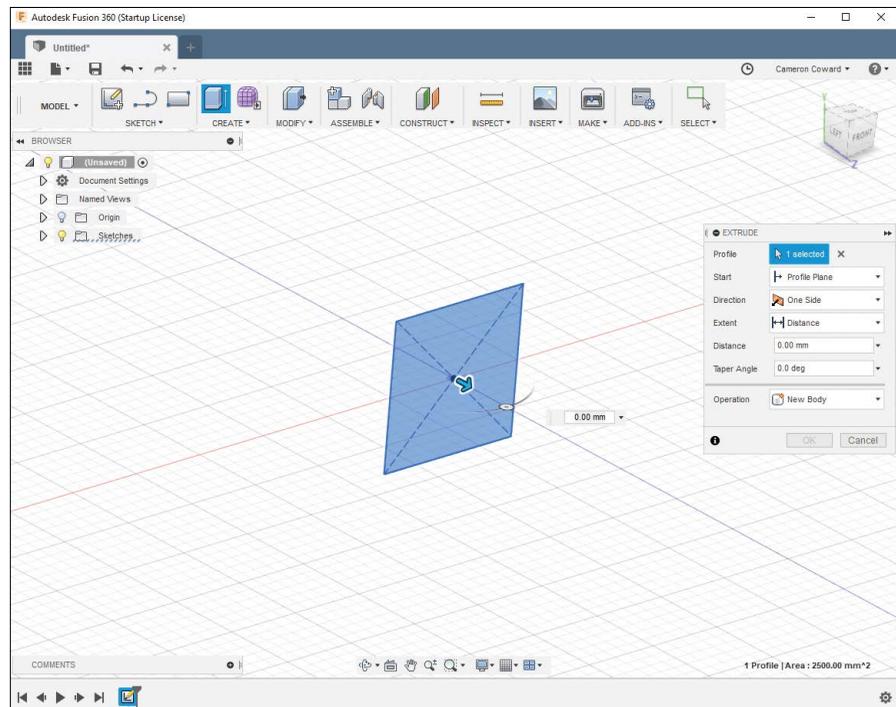

Figure 3-7: The Extrude dialog

The Direction option determines whether the extrusion should extend to one side of the sketch plane, both sides with different distances, or both sides with symmetrical distances. You want your cube to stay centered on the origin point, so choose **Symmetric**. When you select Symmetric, you'll be asked if the dimension refers to total extrusion length or per side. You want total so that all dimensions match. Set the Distance option to 50 mm with a 0 degree taper angle.

The last option, Operation, determines what the feature does. Because you don't have any other solid bodies in your model, New Body is the only option that makes sense, but if you wanted to cut a hole in an existing body, you could use Cut. Other options are Intersect, which keeps just the section where two solids overlap, and Join, which merges two solids together.

Finally, click **OK** to create your very first 3D model! The finished model should look like the cube in Figure 3-8.
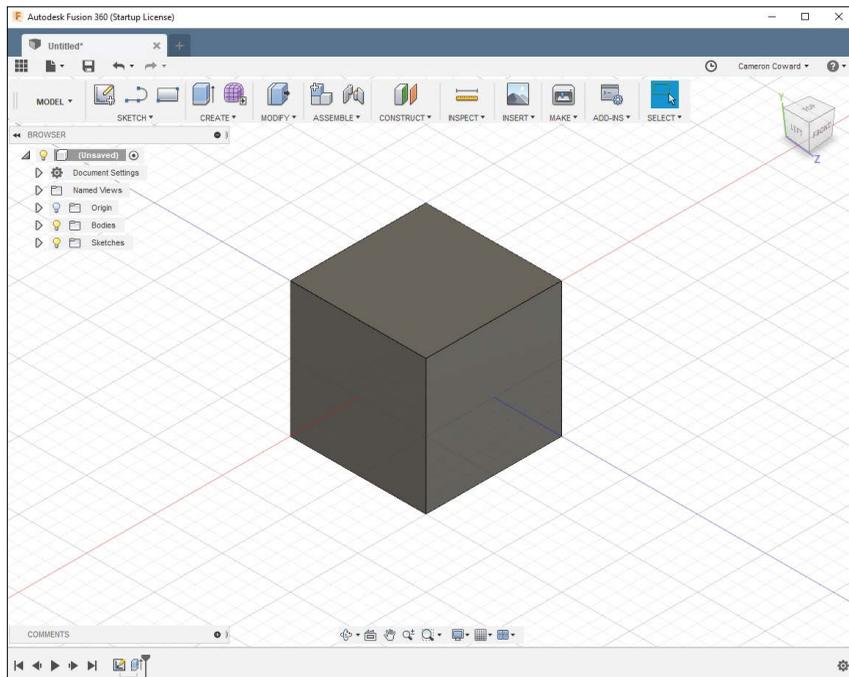

Figure 3-8: A cube, your first 3D model!

Now that you've completed the cube, let's give it some more character with some modifying tools.

Modifying the Cube

In this section, you'll fillet all of the cubes' edges and cut a circular hole into one of its sides. Filleting the edges gives your cube a nice rounded look.

Because Fillet is a modifying feature, you don't have to sketch anything; you only need to select the edges you want to fillet. Be sure to rotate the model to get the back edges and choose a radius of 5 mm, as shown in Figure 3-9.
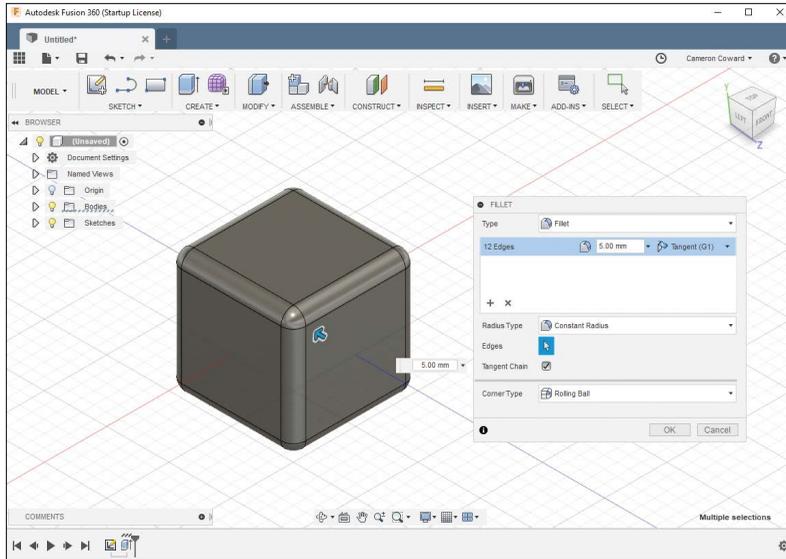
Figure 3-9: Select edges to fillet.

To better understand how the Fillet tool works, try tinkering with the radius measurement and note its behavior.

To cut a hole through one side of the cube, create a new sketch. For the sketch plane, choose either the front or back flat face of the cube (you'll see why soon). Next, sketch a **Center Diameter Circle**, with the center on the origin point and a diameter of 25 mm, as shown in Figure 3-10.
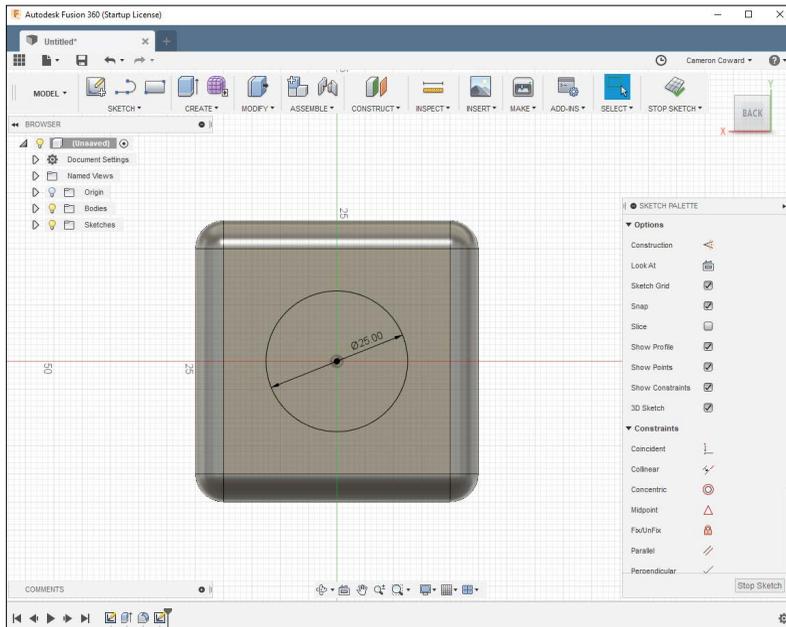

Figure 3-10: Circle drawn with centerpoint on the origin on one side of your cube

This is why you took the time to center your rectangle around the origin and use a symmetric extrusion. When you select a sketch plane, the origin point is automatically centered on your part, because the previous features were modeled symmetrically around the origin. Now when you center your circle on the origin, you know that it's also centered on the face of the cube. If your cube wasn't centered on the origin, you'd have to use additional constraints, construction geometry, or dimensions to center the circle on the face.

With the circle drawn and fully constrained, you can extrude it to cut the hole. To execute a one-sided Extrude, set the Extent option to **All** so that it cuts through all existing solids. You may need to select the **Flip** toggle to change the direction, and then set the operation to **Cut** (see Figure 3-11).
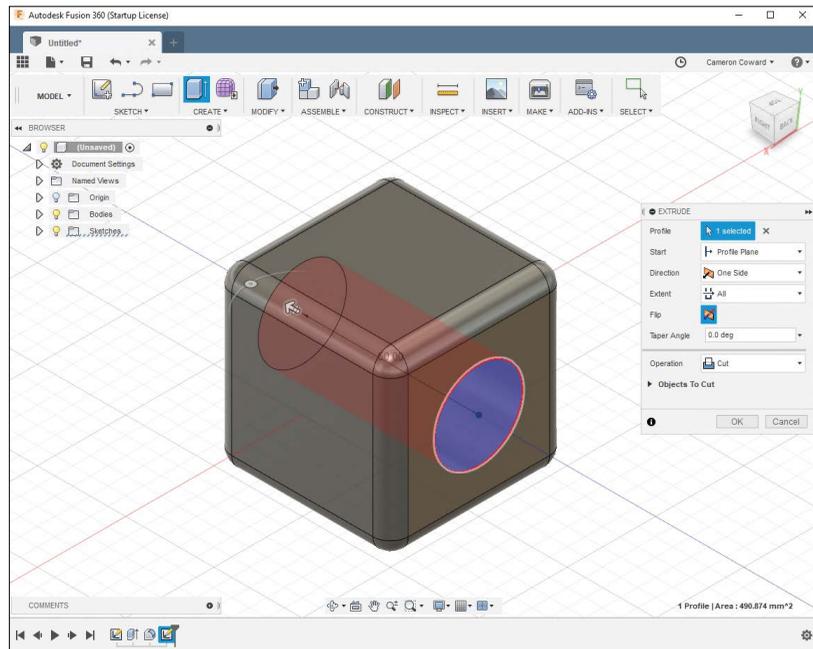

Figure 3-11: Using Extrude to cut a hole

Your cube now has a hole in it—nifty! To save your model, click the **Save** icon.

Using the Design History Timeline

You may have noticed that icons have started appearing at the bottom left of the window in the Design History Timeline. These icons represent each step you've taken to create this particular model (see Figure 3-12).

You can also use the Back and Forward buttons to skip back and forth through all of the features to see each step. You can even use the Play button to see a sort of time lapse of your modeling process.

Being able to skip through the features is cool, but the true power of the Design History Timeline lies in its ability to modify those old features and sketches. When you use the timeline to make changes to an earlier step, all subsequent items in the Timeline should update to reflect your changes. Let's give it a try!

Figure 3-12: The Design History Timeline

Right-click the second item in the Timeline—the first should be a sketch, and the second should be the initial extrusion. Next, click **Edit Feature**, and the original Extrude feature dialog should pop back up. Now change the distance from 50 mm to 100 mm and then click **OK**. The result should look like the model shown in Figure 3-13.
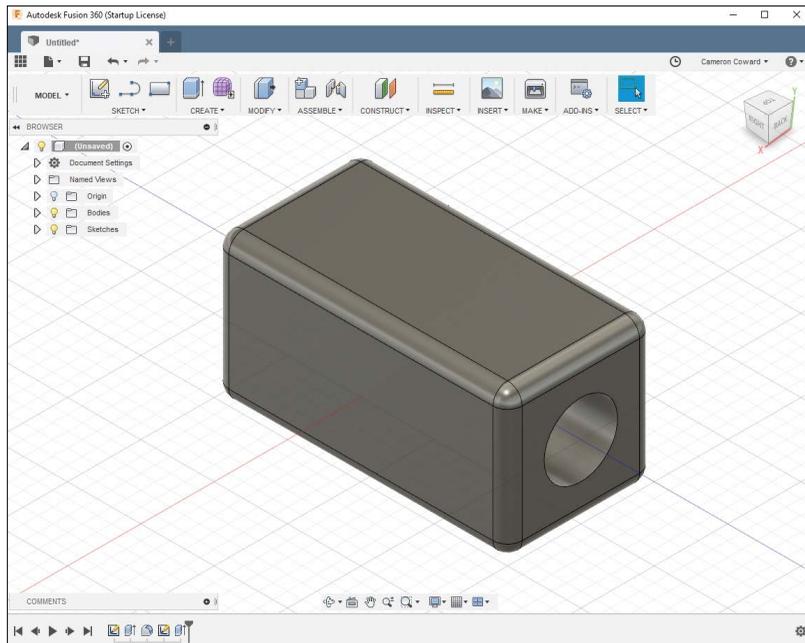

Figure 3-13: All of the features after the change update to reflect the change.

Notice how the edge fillets were lengthened, but the hole still goes all the way through the part. Thanks to the power of parametric modeling, Fusion 360 groups geometry into variable-driven features that adapt to changes. The edge itself had the fillet, so when the edge got longer, the fillet did too. Recall that when you extruded the hole cut, you chose the All option, so when the extrusion was lengthened, so was the cut.

The reason you chose the front or back face for this feature was to avoid an unexpected result, like the one shown in Figure 3-14.
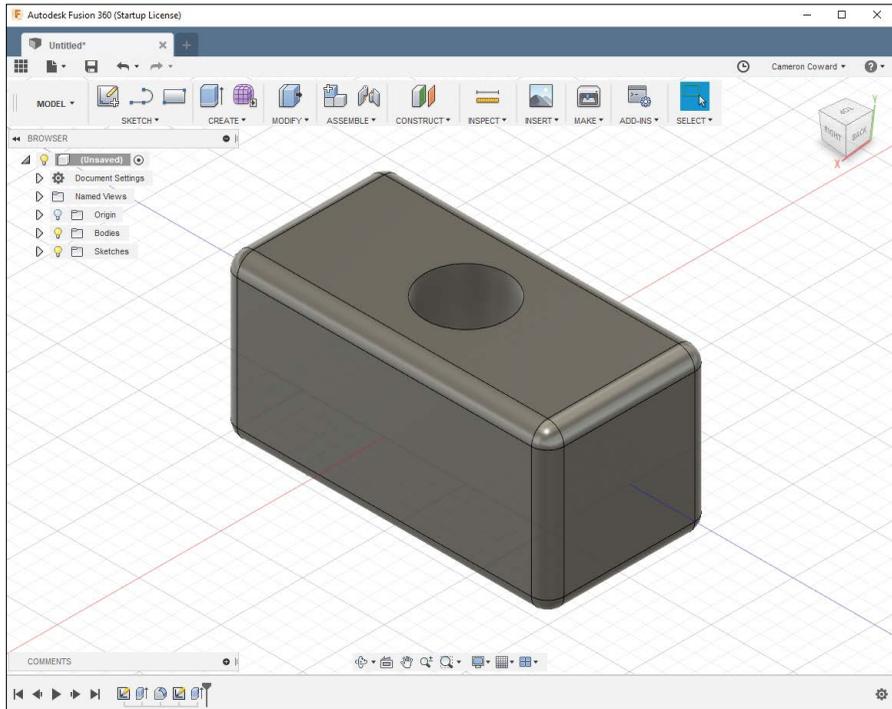

Figure 3-14: The hole is on the wrong face!

In this case, the hole was extruded through one of the side faces. When the model was just a cube, it looked the same no matter which face you sketched the hole on. But when the model was extended, the hole could end up in the wrong place if you chose the left, right, top, or bottom faces. How you choose to orient models is completely up to you, as long as you keep in mind how the features are positioned relative to each other.

Exercises

Practice the techniques you learned in this chapter by completing the following exercises on your own.

Add a Slot

Remove the hole in the block by right-clicking the corresponding feature in the Design History Timeline and selecting **Delete**. In this case, the hole feature is the last one in the Timeline. But, if you ever lose track of the features, you can hover your cursor over the Design History Timeline entry icons to highlight the corresponding feature in the model, or drag the marker back through the Timeline to see each modeling step.

Now you'll add a slot to the top face. To sketch a slot, you can use the Slot tool in the Sketch menu. There are different options for how to define it, and you'll want to use overall length. Give it a length of 50 mm and a width of 20 mm. When you're done, it should look like the model shown in Figure 3-15.
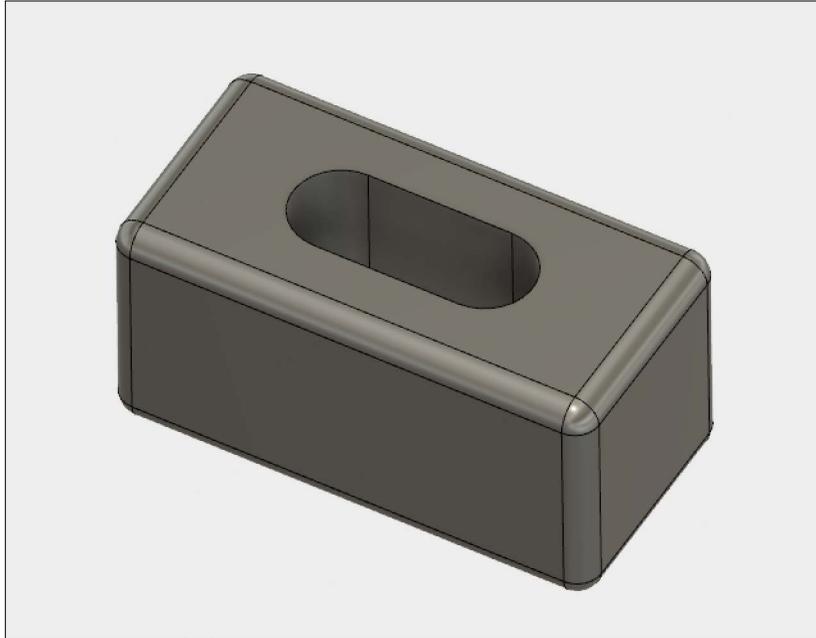

*Figure 3-15: Add a slot to the top face of the block.*

Make the Slot Responsive

As you've already seen, it's important to consider how models will change in the future. In the first exercise, you made the slot 50 mm long—half the length of the block itself. Suppose you *always* want the slot to be half the block's length, no matter what that length is.

To do that, edit the slot sketch. Under the Modify drop-down, click **Change Parameters**, and a dialog will pop up that lists all of dimensions used in the model (you may have to expand the submenus). Parameters are divided into User Parameters, which you explicitly create, and Model Parameters, which Fusion 360 generates automatically when you model new features. One of the dimensions will be the Extrude Distance of 100 mm. Take note of its name, which was "d3" in my case.

Next, you can modify the slot length dimension with that name so it says something like "d3 / 2". Now, the slot length will always be calculated as being half of the extrude distance.

If you elongate the block, your model should look something like the model shown in Figure 3-16.
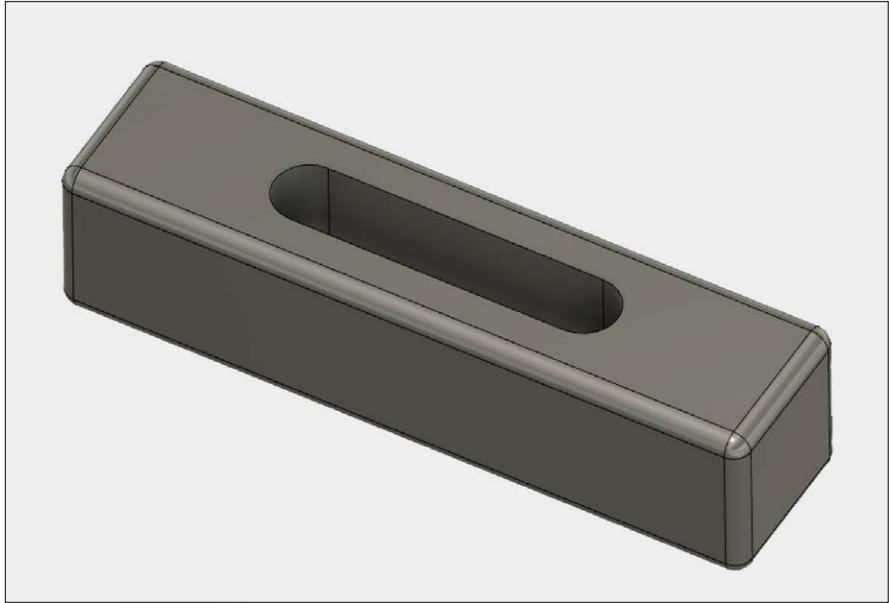
Figure 3-16: Use parameters to make the slot length always half the block length.

Summary

Using a simple example of CAD modeling, you learned how to use Fusion 360's most versatile tool: Extrude. In the following chapters, you'll dive into increasingly complex features, but you'll continue to use Extrude throughout this book and in all of your 3D modeling projects.