Basic Modelling in Fusion 360
Presented by Paul Hermany

This short program is intended to give you an introduction to modelling using a program called Fusion 360. You will learn the basics of parametric modelling (in contrast to direct modelling), learn how to set up a project in Fusion 360, and create a simple object for 3d printing.

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Meet your Serial Hobbyist

My name is Paul, I'm a father of 4, an IT professional, and a self-proclaimed serial hobbyist.

My favorite hobbies include music, tabletop gaming, gardening, painting, woodworking, cooking, and of course, **3d modelling and printing**.

What is Parametric Modelling?

Parametric modelling is a modelling process where 3d objects are created based on formulas, dimensions, and constraints rather than direct manipulation. With direct manipulation, using a program like Tinkercad or Meshmixer, 3d objects are placed in a virtual workplane then manipulated using simple operations such as join, split, solid, and hole. If you need to change the size or shape of the object, those simple operations must be performed again. By contrast, using parametric modelling, you define the shape with parameters such as size, diameter, angle, and

extent. If you need to change the size or shape of the object, the parameters can be adjusted and the resulting object is re-rendered automatically.

In the simple example below, let's say our goal is to create a disc with a quarter slice removed. Using Tinkercad in Figure 1, we can add the whole (positive) disc to the workplane followed by a "hole" (negative) cube. By joining the negative cube with the positive disc, the result is the finished object. Using Fusion 360 in Figure 2, we first define a circle with a center point and radius, then we define two lines extending from the center of the circle to the circumference. The final step is to extrude the flat shape into the third dimension.
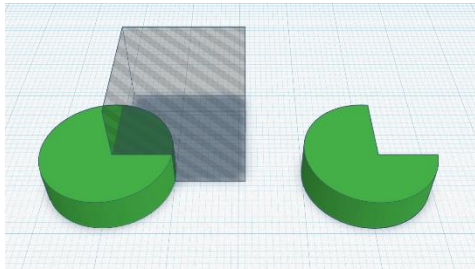

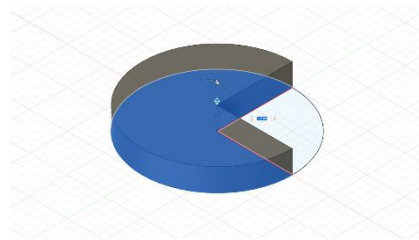*Figure 1: Joining a solid disc with a square hole to cut the disc*

Figure 2: Extruding the disc from a sketch.

Direct modelling is quick and highly effective for creating basic "differential" 3d models or models that are a combination of smaller shapes. We can use Tinkercad to create simple designs or prototypes that we can then quickly send to the printer as a proof of concept.

Parametric modelling is the preferred method if we want to design a component with precision, or if we think any of the measurements of the object are likely to change in the future.

Making a Basic Parametric Box

To demonstrate the basics of parametric modelling and how to get started with Fusion 360, follow these steps to model a basic box for 3d printing.

In this example, we're going to create a simple rectangular box using the parametric modelling technique. We'll create a sketch of the top view of the box, then extrude the walls and base of the box into the third dimension.

Step 1: Save the project
LEARN

All the work you do in Fusion 360 is saved as a **document** inside a **project**. When you save the document for the first time, you'll need to create a new project. Subsequent edits to the document will automatically be saved to this new project.

In Figure 3, the project name is "Basic Parametric Box" and the document name is "Untitled".

In the non-commercial version of Fusion 360, you are limited to 10 editable projects at a time. You must change a project to a read only, non-editable state to edit additional projects.
DO

Save the document and name the project anything you want. "Basic Parametric Box" is a reasonable choice.
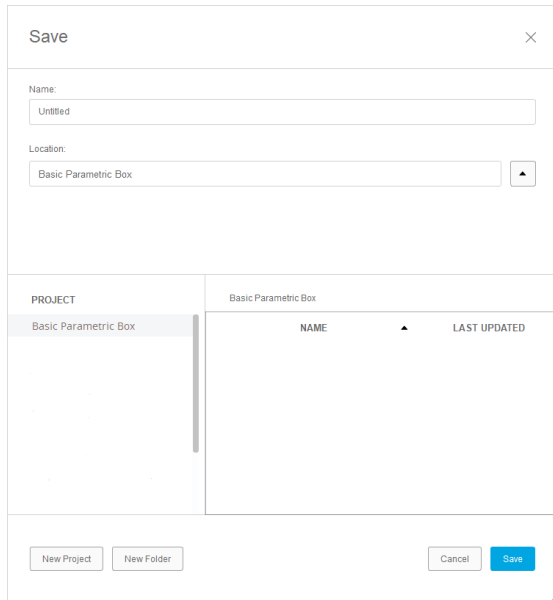

Figure 3: Fusion 360 "Save" dialog

Step 2: Create a sketch
LEARN

When you create a **sketch**, you'll need to select a **plane**. In an empty document, the three basic planes are X:Y, Y:Z, and X:Z corresponding to the planes perpendicular to the Z, X, and Y axes respectively. A plane extends infinitely in both horizontal and vertical directions.

New sketches will appear in the object **browser** to the left.

Sketches can be created on any plane, even planes defined by object geometry. For example, if you create a pyramid shape, one of the faces of the pyramid can be used as the plane for a new sketch.

DO
1. Click the Create Sketch button or select Create a Sketch from the Create menu.
2. Move your mouse near the origin until the X:Y plane is selected as in Figure 4. Click the highlighted panel to create the sketch.
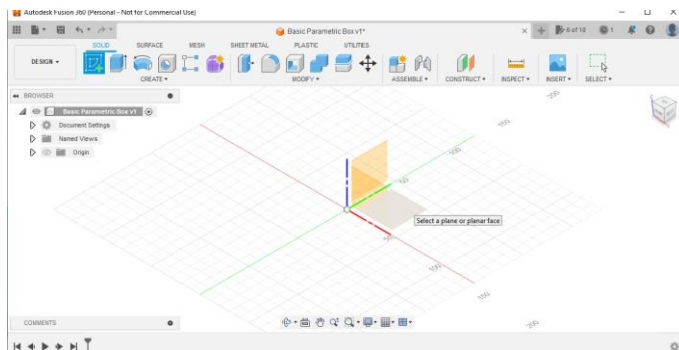

Figure 4: Selecting the X:Y plane

Step 3: Sketch a rectangle

LEARN

In sketch mode, the **create** and **modify** menus will show options relevant to two dimensional sketching (instead of showing options for direct three-dimensional modelling).

To exit the sketch and return to the full document, click the **finish sketch** button highlighted in Figure 5.

While in sketch mode, you can add two-dimensional line drawings to the sketch.
The **linetype** option lets you choose between regular and construction lines. Construction lines are draft lines used for relative positioning and are not part of the final rendering.

DO
1. Click the 2-Point Rectangle button or select 2-Point Rectangle from the Create menu.
2. Click the Origin point.
3. Click any other point on the plane to create the Rectangle.

OBSERVE

Black lines indicate fully constrained lines. These lines are not movable because the geometry forces them into their current position.

Blue lines indicate unconstrained/movable lines. These lines can be resized or otherwise moved within their constraints.

Figure 6 shows a rectangle with two lines which are fully constrained by the position of the second point and their vertical/horizontal orientation. The 2-point rectangle tool automatically adds the vertical/horizontal constraints.
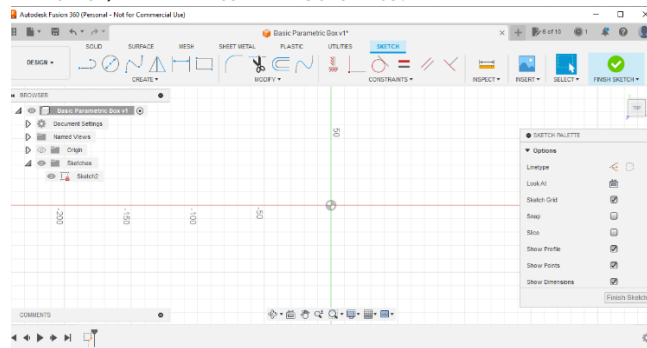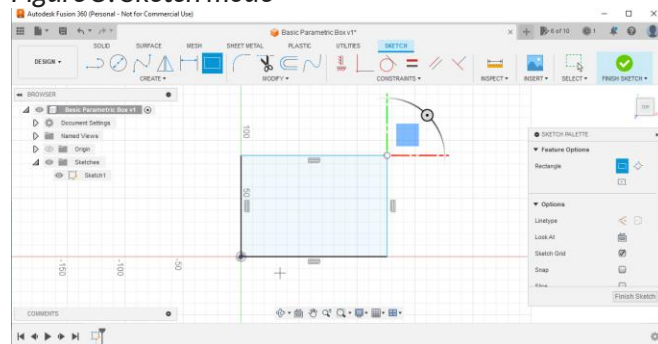

Figure 5: Sketch mode

Figure 6: The partially constrained rectangle

Step 4: Fully constrain the rectangle
LEARN

You can add, remove, and modify **constraints** on line drawings including horizontal/vertical, coincident, perpendicular, equal, parallel, and others. Constraints can be added until the sketch is fully constrained.

Some objects, like rectangles, will automatically include default constraints. Additionally, placing a line on the midpoint of another line or the intersection point of two lines will add a midpoint and coincident constraint respectively.

DO
1. Click the Sketch Dimension button or select Sketch Dimension from the Create menu.
2. Click on the unconstrained (blue) vertical line.
3. Click near the rectangle to place the dimension
4. Repeat to add a dimension to the horizontal line.

OBSERVE

The rectangle is additionally (but not fully) constrained by the first added dimension as seen in Figure 7. Since we chose a fixed height for the rectangle, the previously unconstrained (blue) horizontal line is now constrained (black).

The value of the dimension can be modified at any time.

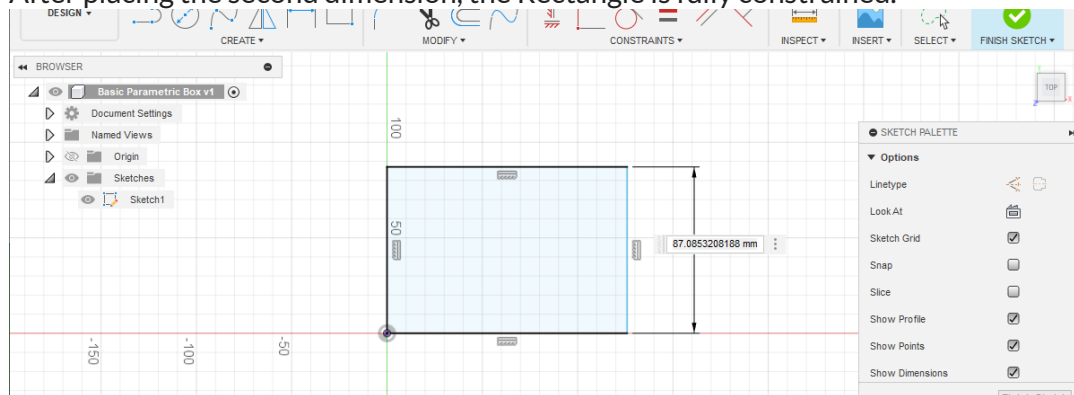After placing the second dimension, the Rectangle is fully constrained.

Figure 7: The rectangle constrained by a dimension

Step 5: Create an offset
LEARN

We've created the outline of the outer wall of the box. To create the inner wall, we'll need to create a second rectangle inside the first. Rather than drawing 4 new lines or creating another 2-point rectangle, we can use the offset tool to automatically copy the rectangle a specified distance from the original lines.

The offset tool will work with any curved or straight lines.

When modelling for 3d prints, I'll typically keep my dimensions multiples of 0.2mm and my wall thicknesses greater than 1.2mm. This will allow walls to be at least 3 layers thick, which is fine for most applications.

DO

1. Click the Offset button or select Offset from the Modify menu.
2. Click the rectangle outline as in Figure 8
3. Drag the slider into the rectangle
4. Set the offset position to the desired wall thickness. -1.2 mm is a reasonable choice.
5. Click Finish Sketch to exit sketch mode and return the main document.

OBSERVE

You can adjust any of the dimensions by double clicking the dimension and typing a new value.

In Figure 9, the rectangle width, height, and wall thickness dimensions are updated with new values: 120mm, 80mm, and 1.2mm respectively.
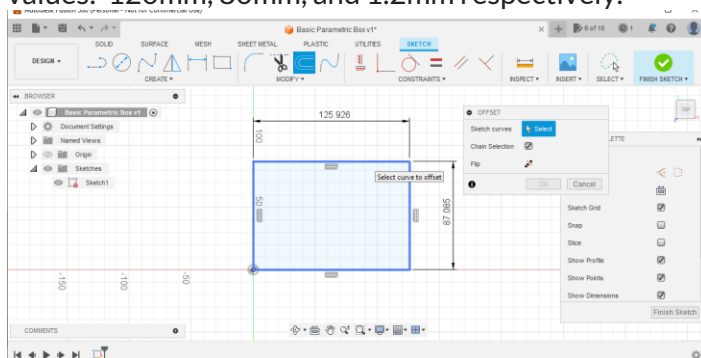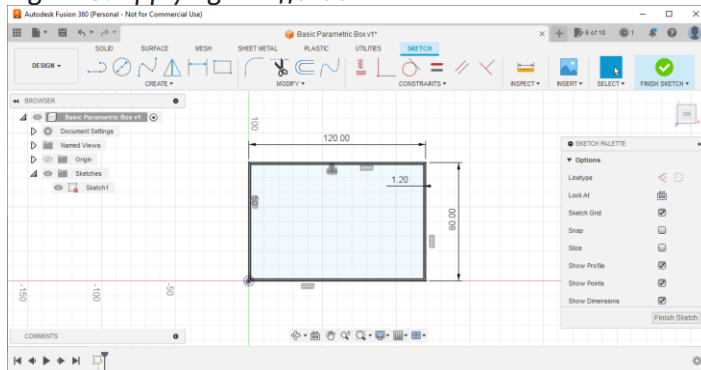

Figure 8: Applying an offset to a curve

Figure 9: The fully constrained rectangle with an offset

Step 6: Extrude the sketch

LEARN

A sketch is a two-dimensional drawing. If we want to continue into the third (Z) dimension, we have to create **bodies**.

You can create a body by selecting an object from the create menu. Alternatively, you can extrude a **profile** or set of profiles from a sketch.

When you extrude a profile, the shape of the profile is extended in a direction perpendicular to the plane. A square can be extruded into a rectangular prism, a circle can be extruded into a cylinder, etc.

DO

1. Click the Extrude button from the Solid menu or select the Extrude option from the Create menu.
2. Select the space between the outer wall and inner wall of the rectangle to choose the wall as in Figure 10.
3. Choose a depth for the box by specifying a distance for the extrusion. 20mm is a reasonable choice.
4. Repeat this process to extrude the base of the box to 1.2mm (wall thickness). Use the join option to join the base to the wall as a single body.

OBSERVE

There are many options to customize the extrusion feature. For example, rather than beginning the extrusion on the sketch plane, you can offset the extrusion so it begins a distance away from the plane. Additionally, you can choose to extrude a profile by distance or until it "hits" another element.

One of the more important customization options to note during extrusion is the **operation**. By default, the new body option is used to extrude the profile into a new 3d dimensional object (body). You can choose to join this body to other connected bodies or choose to cut or intersect the extrusion with existing bodies. This allows for the creation of (among other things) holes in walls.

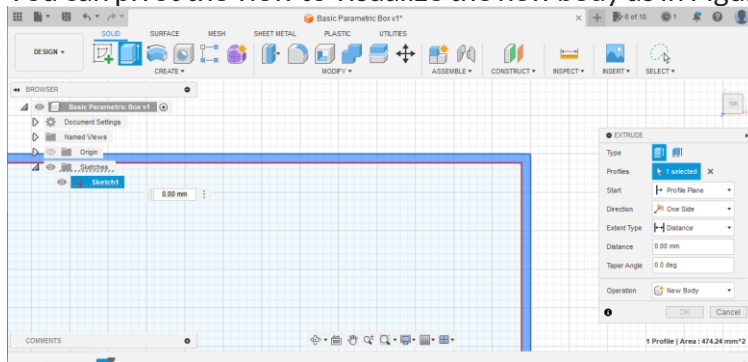You can pivot the view to visualize the new body as in Figure 11.

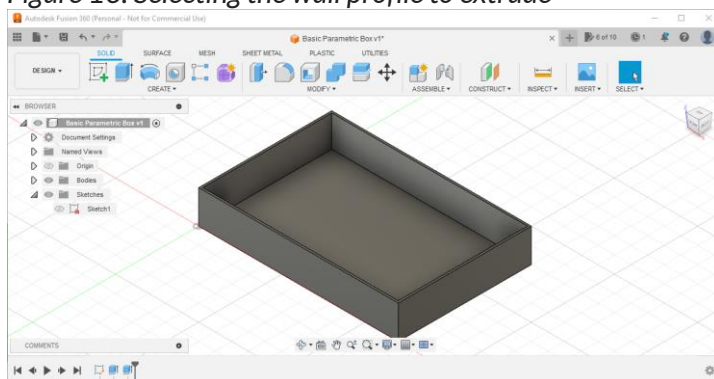Figure 10: Selecting the wall profile to extrude

Figure 11: Viewing the extruded wall and base in three dimensions

Step 7: Export the model
LEARN

Once you've created bodies in three dimensions, you can export them for 3d printing as STL or 3MF files.

STL files store basic mesh data that can be used in a slicing tool to be sent to a 3d printer.

3MF files are a more robust format that contain additional information about the model that can be helpful for more complex print projects.
DO
1. Click the 3d print button from the **utilities** tab or select the 3d print option from the **make** menu.
2. Click the joined base and wall body as the selection as in Figure 12.
3. Uncheck the send to 3d print utility option.
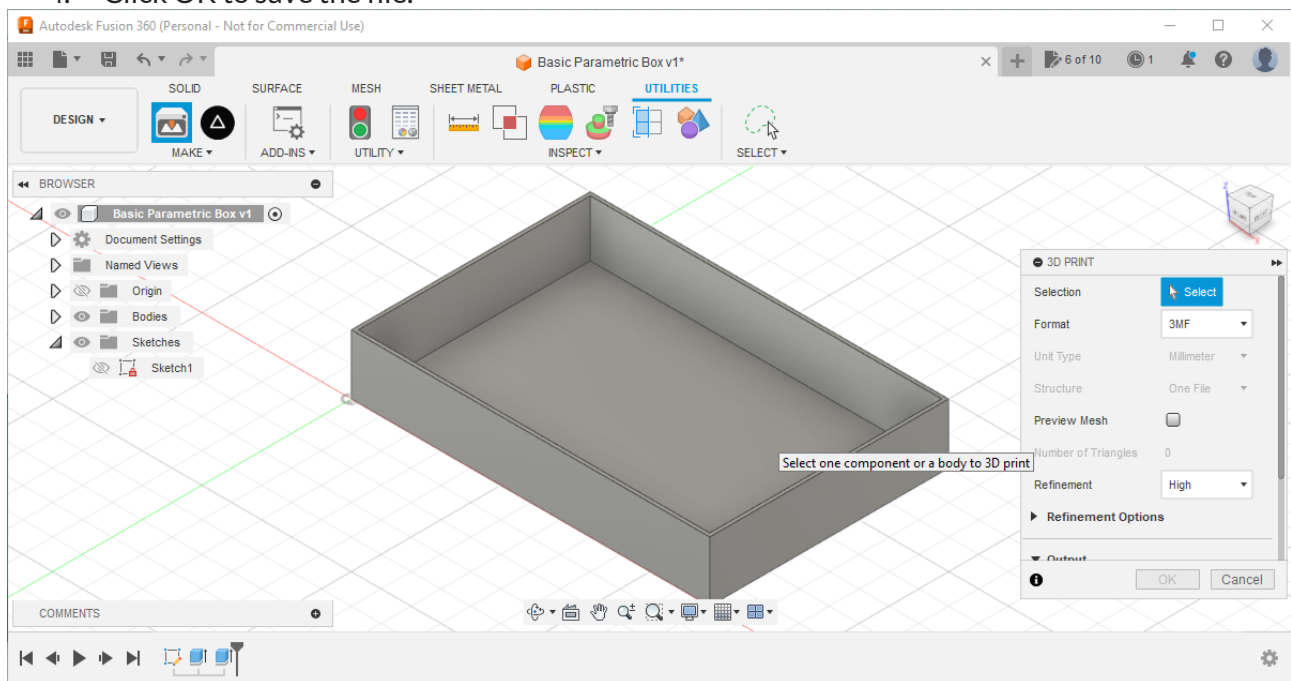4. Click OK to save the file.

Figure 12: Exporting the body as a 3mf file for printing

Making a Playing Card Tray

To demonstrate some additional capabilities of Fusion 360, follow these steps to turn the basic box into a useful tray for playing cards. In this example, we'll start with the basic box, add some custom parameters, then cut a finger hole in the side of the box.

Step 1: Take some measurements
DO

Measure a deck of playing cards

A standard deck of playing cards is appx 2.5" x 3.5" or 63mm x 88mm,

The depth of a standard deck varies by material, but most decks of coated playing cards will stack to 5/8" or 16mm.
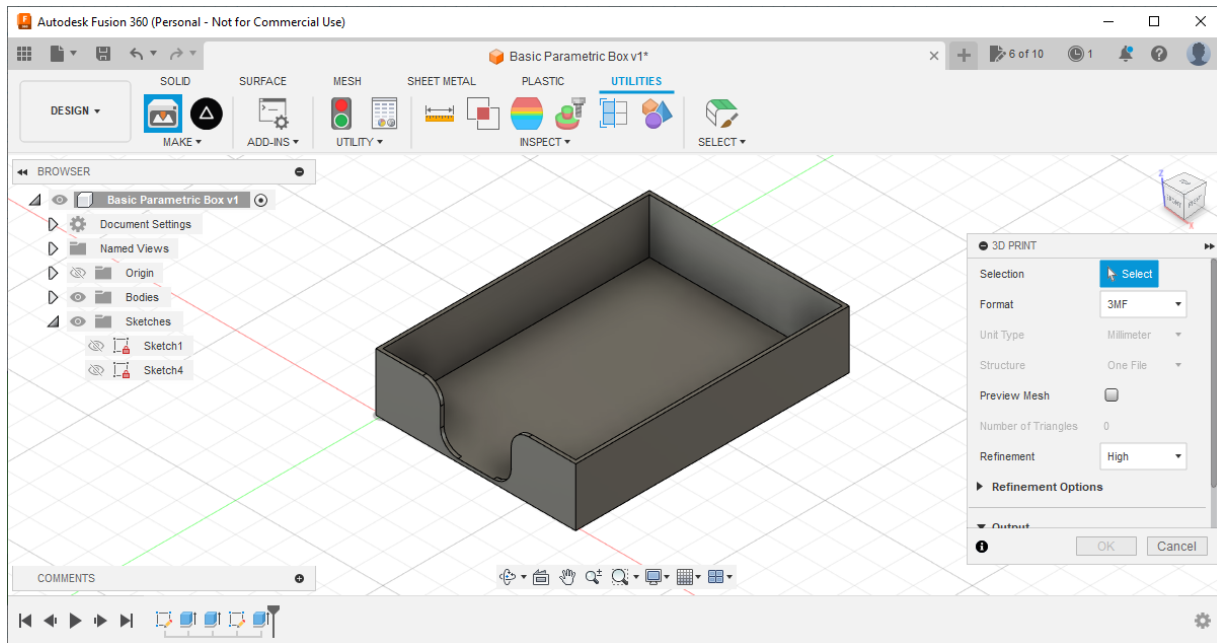

Figure 13: The completed playing card tray

Step 2: Set up custom parameters
LEARN

Fusion 360 will automatically track parameters from the model. Any dimensions or numeric values configured for lines or features (like extrusions) will be stored in the model parameters section as in Figure 14.

You can create custom user parameters to keep track of numbers you may want to reuse throughout the project.
DO
1. Create a user parameter for card width and card height. Set these values to the exact width and height of the card (63mm x 88mm).
2. Create a user parameter for the depth of the deck. Set this value to the approximate depth of the cards at rest with no pressure applied (16mm).
3. Create a user parameter called "tolerance" which will be added to other values as a buffer to account for measurement error, squish, bloat, and other physical anomalies. Set this value to a reasonable tolerance (2.4mm).
4. Create a user parameter for wall thickness and set it to at least 1.2mm.
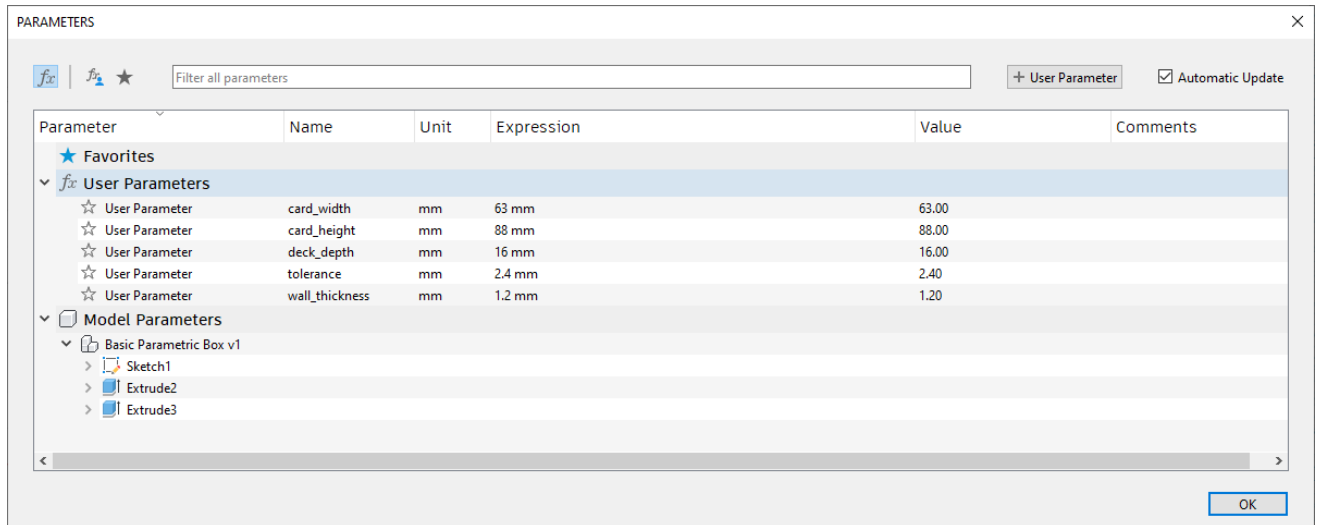
Figure 14: Adding custom user parameters

Step 3: Parameterize the box
LEARN

Any parameters can be used as values for other dimensions and feature settings. For example, if you add a dimension to set a line to 10mm long, you can set another line to be twice the value of that dimension (20mm). It's helpful to try to make the model self-referencing when possible to reduce the amount of changes needed when a single dimension or value needs to be updated.

DO

1. Change the value of the horizontal dimension of the box to be the value of the card width plus the tolerance.
2. Change the value of the vertical dimension of the box to be the value of the card height plus the tolerance.
3. Change the value of the offset wall dimension to be the wall thickness.
4. Change the value of the wall extrusion distance to be the deck depth plus the wall thickness plus the tolerance as in Figure 16.
5. Change the value of the base extrusion distance to be the wall thickness.
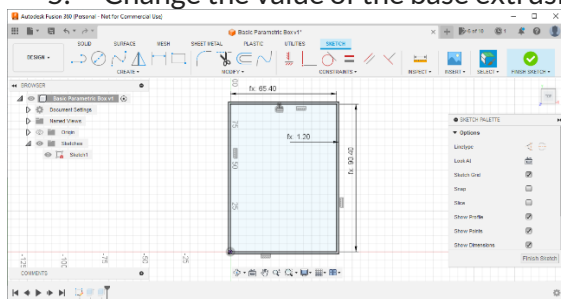

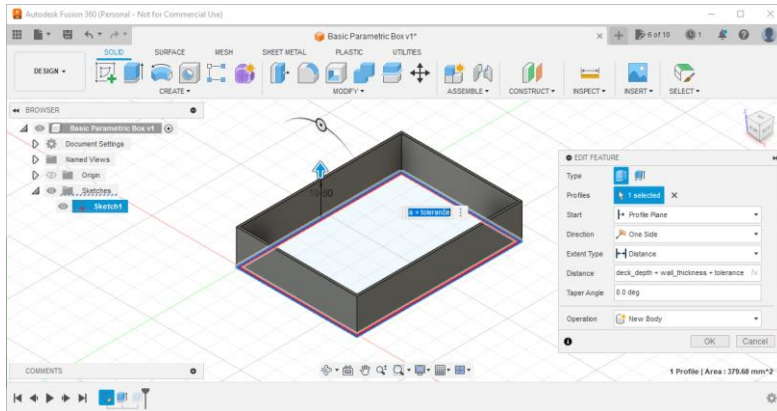*Figure 15: Updating the dimensions of the box with the user parameters*

Figure 16: Updating the extrusion values with the user parameters

Step 4: Create a sketch on an existing body
DO

Create a new sketch by selecting one of the inside shorter walls. Use the inside wall so the sketch projects the top and bottom of the wall excluding the base.
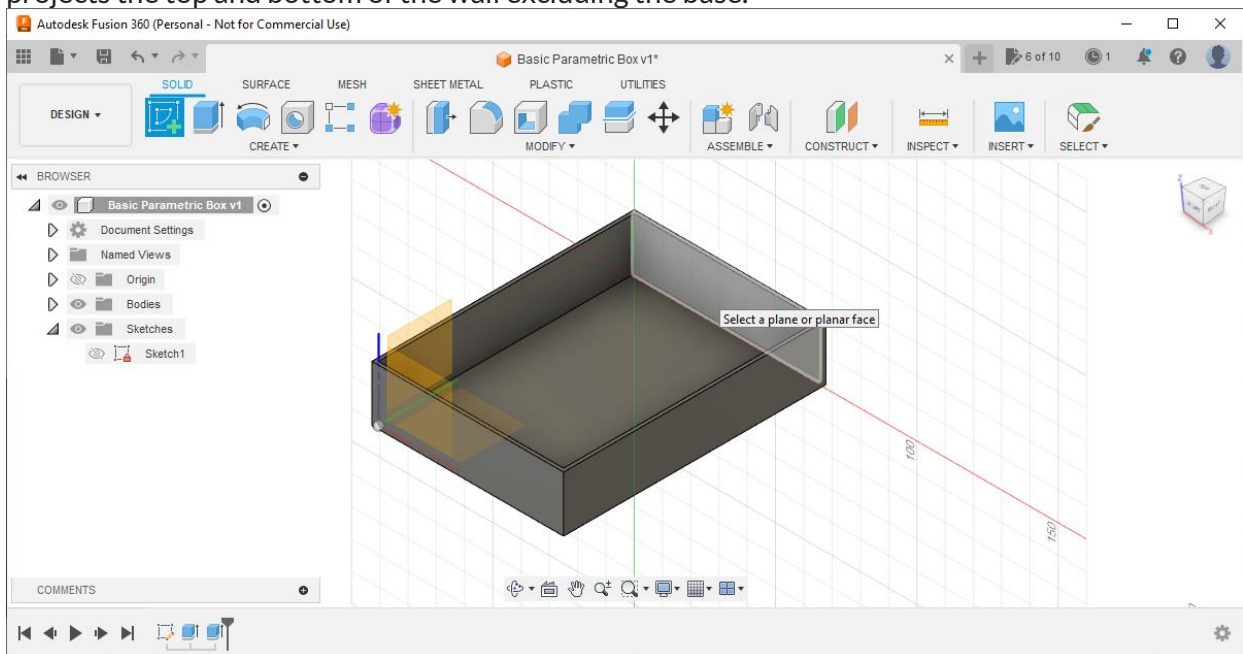

Figure 17: Creating a sketch on an existing body

Step 5: Sketch finger holes
DO
1. Sketch a construction line starting at the midpoint of the sketch as in Figure 18.
2. Following the general design from Figure 19, sketch a line from the top corner to just about halfway to the center line along the top edge, then another line from that point extending perpendicular until it meets the bottom edge, then a third line from that point extending to the center line along the bottom edge.
3. Use the dimension tool to set the distance from the vertical line to the center construction line to be 12mm.

4. Use the fillet tool to make a 6mm rounded corner on the top and the bottom of the three connecting lines.
5. Use the mirror tool to mirror the left-hand side curve on the right-hand side using the center line as the mirror line as in Figure 19.
6. Finish the sketch.

OBSERVE

As you complete the above steps in the sketch, notice how the fillet and mirror tools will automatically add constraints to the sketch. The fillet tool will add a curved line with a tangent constraint and the mirror tool will add symmetry constraints to copies of the lines on the other side of the mirror line.

The same effect can be achieved by creating simple curves and adding constraints. The fillet and mirror tools make the process easier.
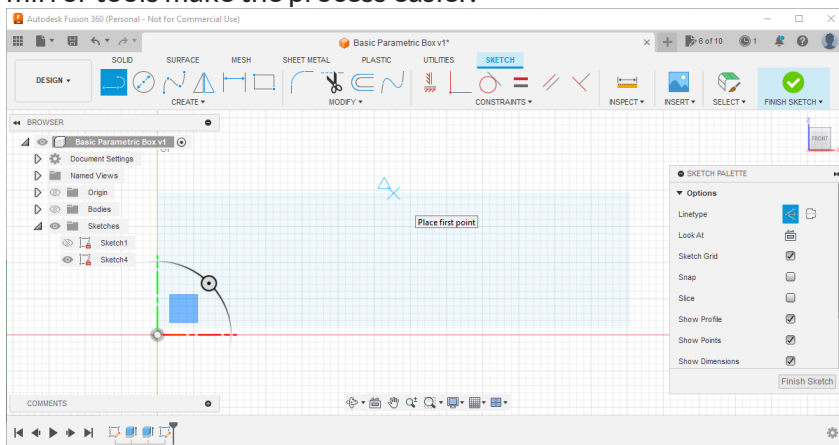

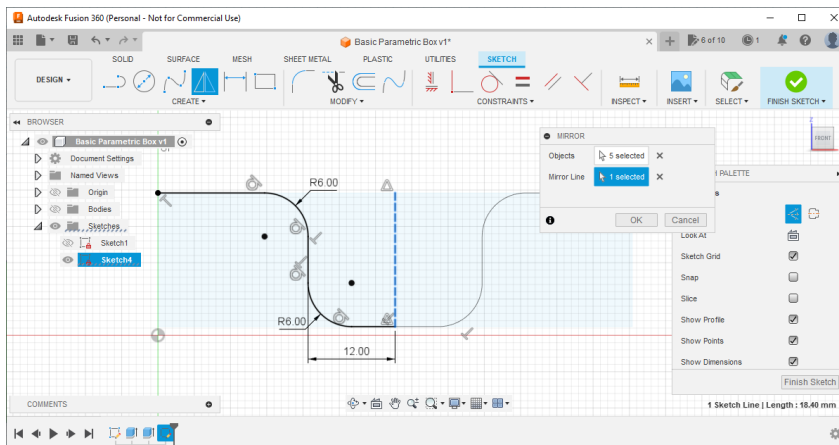*Figure 18: Creating a new construction line at the midpoint of the sketch*

Figure 19: Sketching the outline of the finger hole using lines and fillets

Step 6: Extrude the finger holes

DO

1. Use the extrusion tool to select the profile of the finger hole in the new sketch.
2. Change the operation to cut.
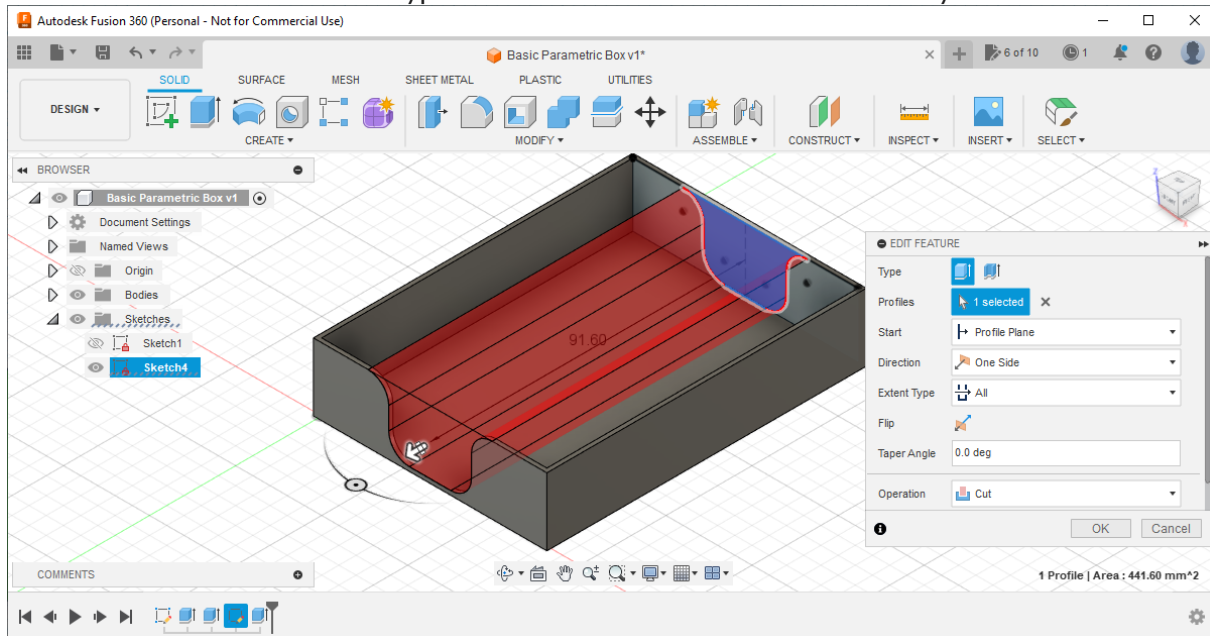3. Select all as the extent type to extend the extrusion to the full body on the one side.

Figure 20: Extruding the finger hole profile as a cut