



FUSION 360: SKETCHING FOR MAKERS





LaDeana Dockery

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MAKEICT Wichita, KS

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Interface

One of the first things that you will want to understand is all the items that are available on the interface.

File Operations

-  Data Panel – Allows you to open existing Projects and create new Projects
-  New – Allows you to create new Designs and other file type operations
-  Save the current Design
-  Undo and Redo

Opening Existing Models

The Data Panel icon will open which would allow you to create New Projects, open previous projects or open shared projects. (Your instructor will collect your email address so that you may open products that are shared for the class.) You can close this panel to get more screen space.

Try it: Opening a Design

Let's open a model for some experimentation:

1. Open the Data Panel (if it is not already open).
2. Open the MakeICT Fusion 360 Sketching for Makers Project (Double-click)
3. Open the Starter Models folder (Double-click)
4. Open **1. Open, view and navigate** (Double-click)

Mouse Navigation

Pan	Hold Middle Mouse Button
Zoom	Roll Middle Mouse Button

Try it: Mouse Navigation

1. Move the sketch up and down
2. Move the sketch left and right
3. Zoom closer and farther away from the sketch

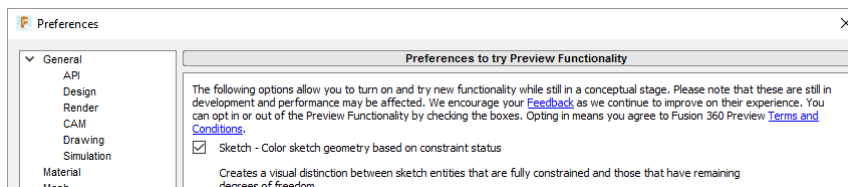
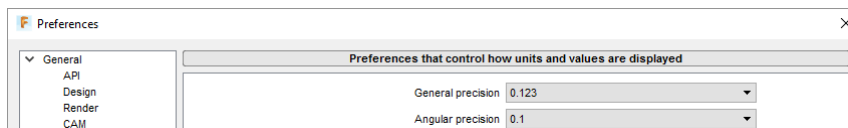
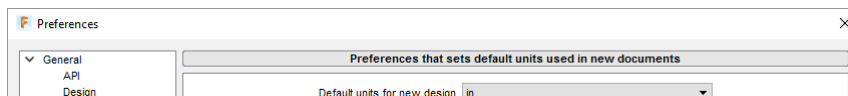
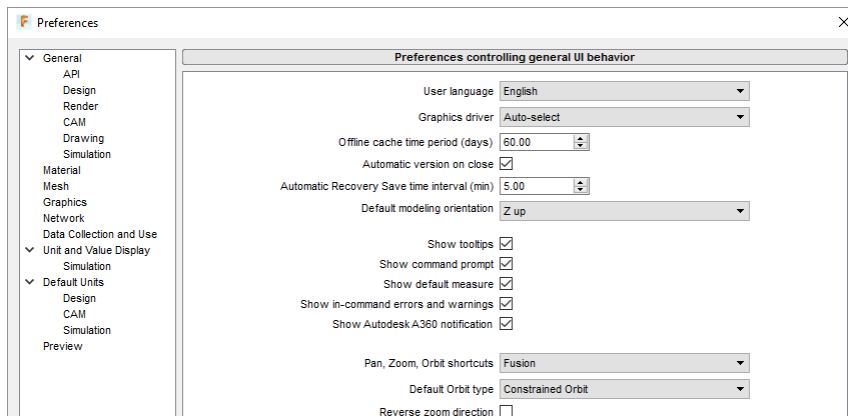
Preferences

There are several individual preferences that you might like to set for yourself. One of them is zoom direction. To set individual preferences:

1. Pick your name from the upper right corner LaDeana Dockery ▾ ? ▾
2. Choose **Preferences** from the menu
3. Choose the preferences you wish to change

A few of the things you might consider changing that are related to this class are

- Save time interval – controls the time that Fusion 360 is set to automatically save
- Default modeling orientation – Z up is what most 3D programs will use
- Zoom direction – if zoom seems backwards to you
- Default units for new design- inches or mm
- Unit precision – how many decimals do you need
- Color Sketch Geometry on Constraint status –turns fully constrained sketch black











Try it: Changing Preferences

Let's change a couple of the options:

1. Change Modeling Orientation to Z-up
2. Put a check on the Sketch – Color sketch geometry based on constraint status

Navigation Bar

The items on this toolbar allow you to view and navigate around the model. The bolded items are the ones to take note of as they are used with sketches.

-  Rotates the model (not needed in sketches typically)
-  **Look at (you can use this if you accidentally rotate out of the plane of your sketch)**
-  Allows you pan the model (Pressing the mouse wheel is easier, but if you forget that...)
-  Zooms the model (Rolling the mouse wheel is easier, but if you forget...)
-  **Fits the model into the view. F6 is the shortcut for this.** (Underneath this icon there is another that allows you to select an area to zoom to)
-  3D Visualization options (not used with sketches)
-  **Grid options – snapping to the grid and setting up grid spacing**
-  Multi-views (not used with sketches)

Try it: Navigation Bar

1. Use the Look At icon to look squarely at the sketch
2. Use the Fit icon (or F6) to fit the sketch on the screen (you may want to move the sketch a little off the screen before you try this to make it easier to see)
3. Check the grid option to make sure that the grid is on and snap is on. Check to see what the grid spacing is currently set to

View Cube

The View Cube allows you to see what orientation of the parts with relationship to the axis and allows you to change that orientation in an intuitive way:

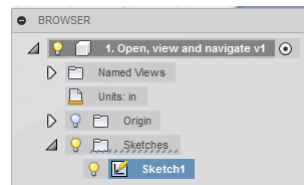
- Pick the side of the cube to look at the corresponding side of the part
- Pick an edge to look perpendicular to that edge
- Pick a vertex to look from that point
- Pick home to go to the home orientation (isometric)
- Right click for more options including setting a new home and going into a perspective view

Try it: View Cube

1. Choose some of the faces, points and edges until you understand what it is doing
2. Select Home when you are done

Browser

The Browser window shows an outline of all the elements in the model. It is sort of the Table of Contents of the model. Right clicking will get you a menu of actions on the item. Double-clicking will make it the active item. The important points for this class are bolded.



- **Name of Design (in our case this is 1. Open, View and Navigate).**
- Named Views – Allows you to create views that you can return to.
- **Units – Units of measure for the model. You can change the units for the model by hovering over it and clicking the icon that appears.**
- Origin – Contains the planes, axes and origin that come with the model.
- **Sketches – Contains all the sketches in the model. Click to open the list. Double-click a sketch to make it the active sketch.**

Try it: Editing a Sketch

1. Double click Sketch1 to make it active
2. Grab the corner of the rectangle and move it (use the left mouse button)
3. Go to the Browser window and select the light bulb. Your sketch geometry should disappear.
4. Turn the sketch geometry back on by clicking on the light bulb again.

Timeline














The order in which you create things matters in FUSION 360. This allows you to go back in time to alter an earlier fact in history and let the changes play out in the model. You will see in your exercises that I created the sketches in order and they will show up in your timeline in that order. When you double click to edit one of them, the subsequent sketches will disappear temporarily.



Toolbar

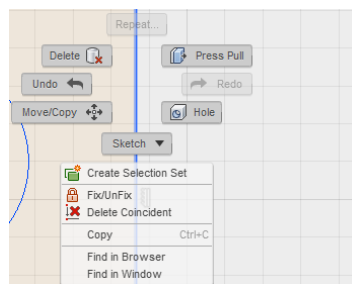


Tools on the modeling workbench are used to model parts. Items relevant to sketching are bolded.

-  **New Sketch – Creates a new sketch.**
-  **Line – Creates a profile of lines and curves.**
-  **Two Point Rectangle – Creates a rectangle from two diagonal points.**
-  **Sketch menu – Host of sketch creation and modification options.**
-  **Create – Creates solid parts.**
-  **Modify – Modifies solid parts.**
-  **Assemble – Assembles solid models.**
-  **Construct – Wireframe construction geometry like planes.**
-  **Inspect – Measure dimensions of your sketch or model.**
-  **Insert – Insert things into a model or sketch like SVGs and DXFs.**
-  **Make – Allows you to send things off to be made by a 3rd party.**
-  **Add-ins – Allows you to use 3rd party enhancements like the sketch gap checking tool.**
-  **Select – Allows different options for selection.**
-  **Stop Sketch – Stops editing the sketch. Only shows up when you are editing a sketch.**

Contextual (Marking) Menu


When you right click over any piece of geometry, a contextual menu appears Autodesk calls this the Marking Menu. Here is a picture of the Marking Menu when I right click over a rectangle



Try it: Toolbar

1. Select the Two Corner Rectangle icon and make a rectangle
2. Right click your rectangle to see the Contextual (Marking) Menu for the rectangle
3. Exit the sketch using the Stop Sketch icon

Help

Often how to do something isn't obvious. Sometimes it is useful to be able to get help from the software. The  will allow you to get help, access training and ask questions on the forum. Notice it pops up a browser window and you must click on Fusion 360 again when you want to make it active.

Try it: Help

1. Select the help icon
2. Key Rectangle into the search icon
3. Select the first result on how to create rectangles
4. If you have time, you can view the short video

Creating Projects, Designs and Sketches

Create a new project (If necessary)

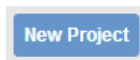
1. Select the Show Data icon, if necessary



2. Select the < icon, if necessary




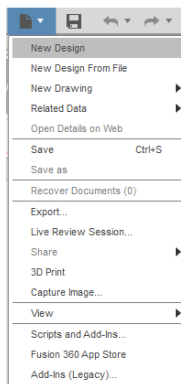
3. Pick New Project




4. Type your project name

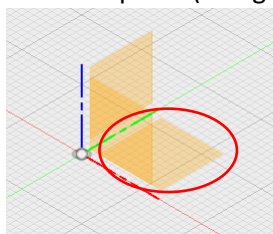
Create a new document

1. Select the New Document icon 
2. Select the New Design menu item




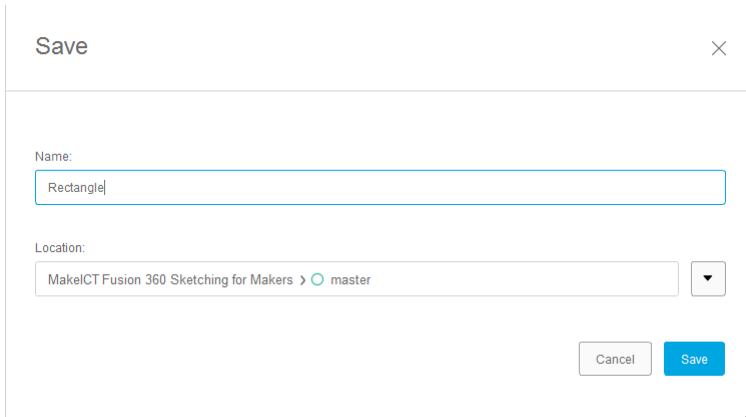
Creating a New Sketch

1. Select the New Sketch icon 
2. Select the XY plane (along the green and red axes)



Save a Document

1. Select the Save icon 
2. Type the Name of the Document and make sure the location is correct
3. Pick the Save button



Save

Name:

Rectangle

Location:

MakeICT Fusion 360 Sketching for Makers > master

Cancel Save

Try it: Creating a new Project, a New Design and a New Sketch and saving them

1. Open the Data Panel (if it is not already open).
2. Create a new project, name it Training
3. Create a New Design
4. Create a new sketch
5. Create a rectangle
6. Save the New Design as “First Sketch”

Copying a Design from an existing project

We will copy several models from existing models to get you some easy geometry to work with. At some point, you might want to copy models from someone else or a library location.

To copy a Design from another person:

1. Open the Data Panel (if it is not already open).
2. Find the Project you wish to copy from
3. Select the folder you wish to copy from (if necessary)
4. Right click the design to copy and select copy
5. Find the location you want to save it
6. Press the Save button

Try it: Copy the Designs for the class

1. Open the Data Panel (if it is not already open).
2. Open the MakeICT Fusion 360 Sketching for Makers Project (Double-click)
3. Open the Starter Models folder (Double-click)
4. Select all the models in this folder
5. Right click the design and select copy
6. Navigate to your Training folder
7. Press the Save button

Creating Basic Profiles

Points

Points aren't really a basic profile, but they are used sometimes for basic profiles. Typical uses for points are to establish midpoints of elements or distances along an element.

To create points:

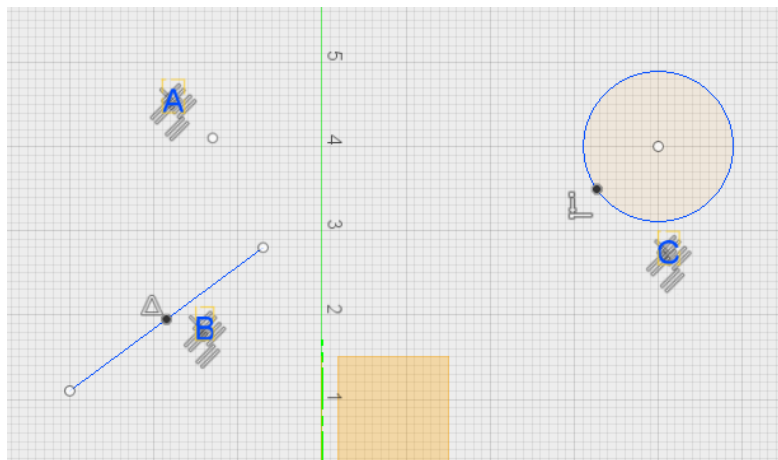
Select a location on the page to snap to

Or

Select a location on the page to be constrained (like a center point triangle)

Try it: Opening an existing project and creating points

1. Open **2. Simple Sketching** (Double-click) from your Training folder (If it is not there, try doing the previous exercise again or get your instructor to help.)
2. Open the Sketches from the browser window
3. Double-Click the Points sketch to edit it
4. Turn on the Points Sketch by selecting the light bulb (If you can't see anything)
5. Create a point near point A (Just pick Point from the menu and click near A)
6. Create a point at the center point of Line B by hovering near the center of the line and picking when the blue triangle appears. (This is a midpoint constraint and will ensure the point is at the center of the line. We will talk more on constraints later.)
7. Create a point on the edge of the circle by hovering over the perimeter of the circle and then selecting when the blue x shows. (The blue x is a constraint meaning that the point must lie on the perimeter of the circle... it is co-incident with the circle. More on this later.)
8. Stop the sketch
9. Turn off the sketch (if needed)
10. Save the design



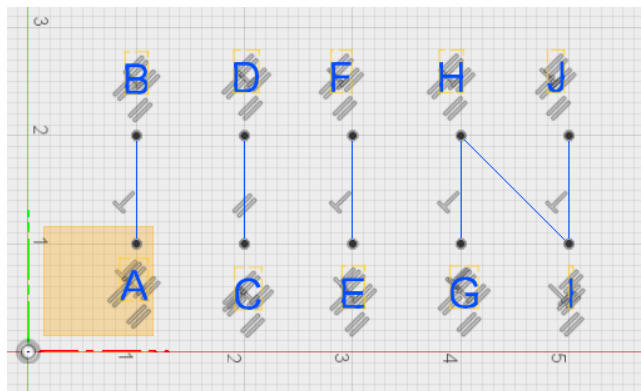
Line (L)

There are a lot of ways to create lines. We will learn the basics here and more in a later section when we discuss profiles

1. Select a point (or snapping location)
2. Select a second point or snapping location
3. Select the check mark or hit the escape key (Esc) or double click the last point to end line creation (or continue picking points until you don't want connected lines)

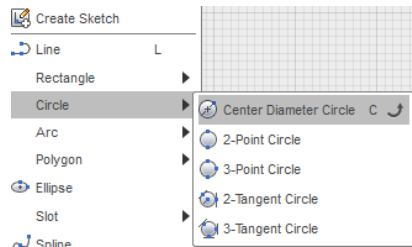
Try it: Creating lines

1. Open the MakeICT Fusion 360 Sketching for Makers Project (If not already open)
2. Open the Starter Models folder (If not already open)
3. Open **2. Simple Sketching** (If not already open)
4. Open the Sketches from the browser window (If not already open)
5. Double-Click the Lines sketch to edit it
6. If you cannot see anything in the sketch, pick the lightbulb
7. Sketch>Lines
8. Pick point A
9. Pick point B
10. Select the check mark
11. Pick point C
12. Pick Point D
13. Hit the Esc key
14. Pick Point E
15. Double-click point F
16. Pick point G
17. Pick point H
18. Pick point I
19. Double-click point J
20. Select the Stop Sketch button
21. Turn off the sketch (if needed)
22. Save the design



Circles

Lines and points only had a single option for each. When you select the circle menu you will find 5 options hiding underneath. Each option will create a circle, you will choose the option you need based on the geometry that you have available to create the circle:



- Center Diameter Circle C – Needs a center and a diameter
- 2-Point Circle – Needs two points on the circumference of the circle directly opposite one another (on the endpoints of the diameter)
- 3-Point Circles – Needs 3 points on the circumference
- 2-Tangent Circles – Needs 2 things to be tangent to and a radius
- 3-Tangent Circle – needs 3 things to be tangent to

Notice that there is a C at the end of the Center Diameter circle option. That is a shortcut to get to that option directly by just pressing the C key. (You also want to notice that there is an L shortcut to get directly to the line function, but no shortcut to get to point. Shortcuts only get made for items that are used a lot... pay close attention to any function that has a shortcut next to it.)

Center Diameter Circle (C)

1. Pick a center point
2. Key the diameter

2-Point Circle

1. Pick a point on the circumference
2. Pick another point on the circumference opposite the first point

3-Point Circles

1. Pick a point on the circumference
2. Pick another point on the circumference
3. Pick a 3rd point on the circumference

2-Tangent Circles

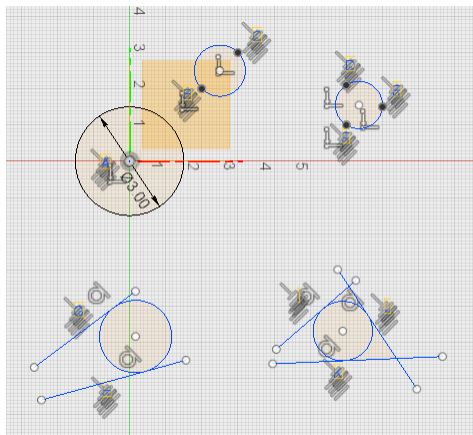
1. Pick an element (like a circle or line) to be tangent to
2. Pick an element (like a circle or line) to be tangent to
3. Key a radius

3-Tangent Circle

4. Pick an element (like a circle or line) to be tangent to
5. Pick an element (like a circle or line) to be tangent to
6. Pick an element (like a circle or line) to be tangent to

Try it: Creating circles

1. Open the MakeICT Fusion 360 Sketching for Makers Project (If not already open)
2. Open the Starter Models folder (If not already open)
3. Open 2. Simple Sketching (If not already open)
4. Open the Sketches from the browser window (If not already open)
5. Double-Click the Circles sketch to edit it
6. If you cannot see anything in the sketch, pick the lightbulb
7. Sketch>Circles>Center Diameter Circle
8. Pick point A
9. Key 3
10. Sketch>Circles>2-Point Circle
11. Pick point B
12. Pick Point C
13. Sketch>Circles>3-Point Circle
14. Pick Point D
15. Pick point E
16. Pick point F
17. Sketch>Circles>2-Tangent Circle
18. Pick line G
19. Pick line H
20. Key 1 for the radius
21. Sketch>Circles>3-Tangent Circle
22. Pick line I
23. Pick line J
24. Pick line K
25. Try a few circles of your own
26. Select the Stop Sketch button
27. Turn off the sketch (if needed)
28. Save the design



Arcs

Arcs are just parts of a circle. Like the circle, there are many choices for making arcs depending on the geometry you have in your model.

3-Point Arc

To make this kind of arc, you need a start and end point and a point on the perimeter of the circle. Notice that the arc will go counter-clockwise from the start point to the end point.

1. Pick start point
2. Pick end point
3. Pick point on the circumference

Center Point Arc

This arc is made from a center point, a start point and an endpoint (or its projection on the circle.)

1. Pick the center point
2. Pick the start point
3. Pick the end point (or a point to project as an end point.)

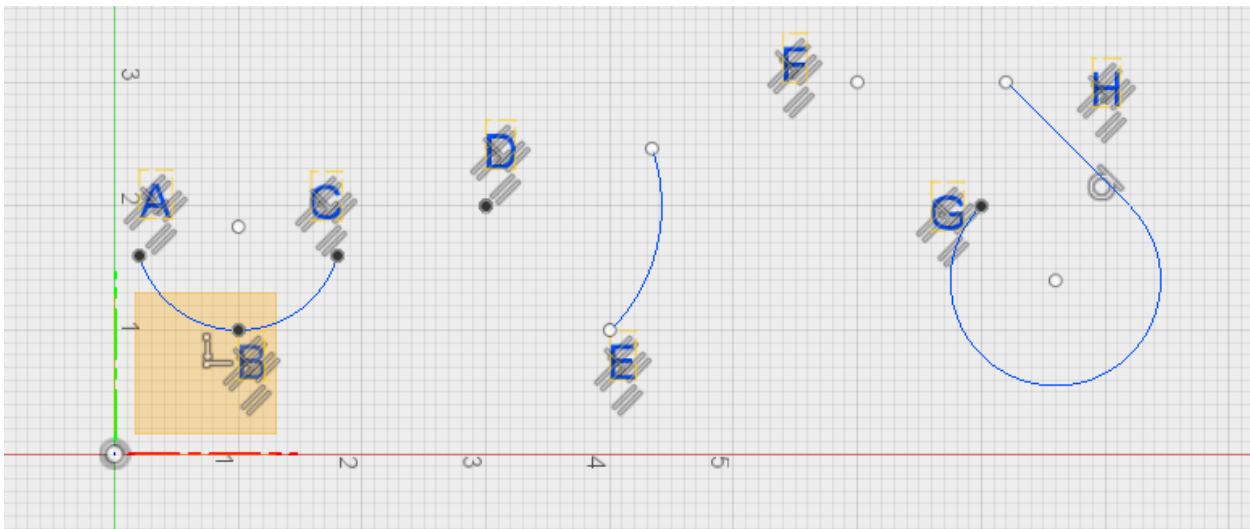
Tangent Arc

This arc is tangent to an element and through a point.

1. Pick the endpoint of an element to be tangent to
2. Pick the other endpoint

Try it: Creating Arcs

1. Open the MakeICT Fusion 360 Sketching for Makers Project (If not already open)
2. Open the Starter Models folder (If not already open)
3. Open 2. Simple Sketching (If not already open)
4. Open the Sketches from the browser window (If not already open)
5. Double-Click the Arc sketch to edit it
6. If you cannot see anything in the sketch, pick the lightbulb
7. Sketch>Arc>2-Point Arc
8. Pick point A
9. Pick point C
10. Pick point B
11. Sketch>Arc>Center Point Arc
12. Pick point D
13. Pick point E
14. Pick Point F
15. Sketch>Arc>Tangent Arc
16. Pick the bottom point of line H
17. Pick point G
18. Try a few arcs of your own
19. Select the Stop Sketch button
20. Turn off the sketch (if needed)
21. Save the design



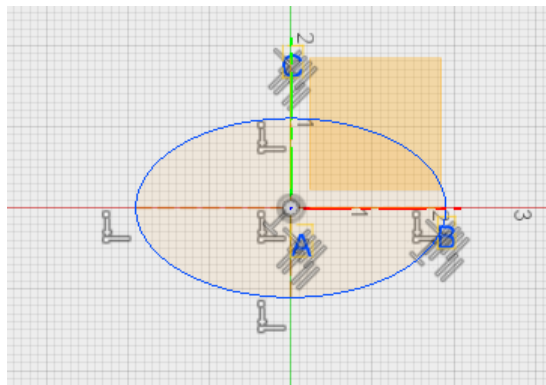
Ellipses

Ellipses (or ovals) have only one option:

1. Select the center point
2. Select a point on the circumference on one of the axes (like a point at the “pointy end” of the ellipse)
3. Pick a circumference point

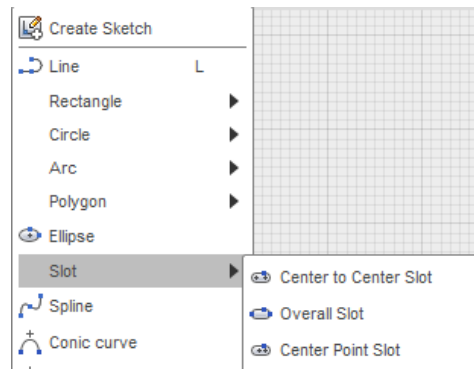
Try it: Creating Ellipses

1. Open the MakeICT Fusion 360 Sketching for Makers Project (If not already open)
2. Open the Starter Models folder (If not already open)
3. Open **2. Simple Sketching** (If not already open)
4. Open the Sketches from the browser window (If not already open)
5. Double-Click the Ellipses sketch to edit it
6. If you cannot see anything in the sketch, pick the lightbulb
7. Sketch>Ellipse
8. Pick point A
9. Pick point B
10. Pick point C
11. Try a few ellipses of your own
12. Select the Stop Sketch button
13. Turn off the sketch (if needed)
14. Save the design



Slots

A slot is two circles and the material between them. You could create these yourself by using circles, lines and trimming operations, but they are used enough that there is a special tool just to build them.



Center to Center Slot

1. Pick one circle center
2. Pick another circle center
3. Pick a point on the perimeter or key width

Overall Slot

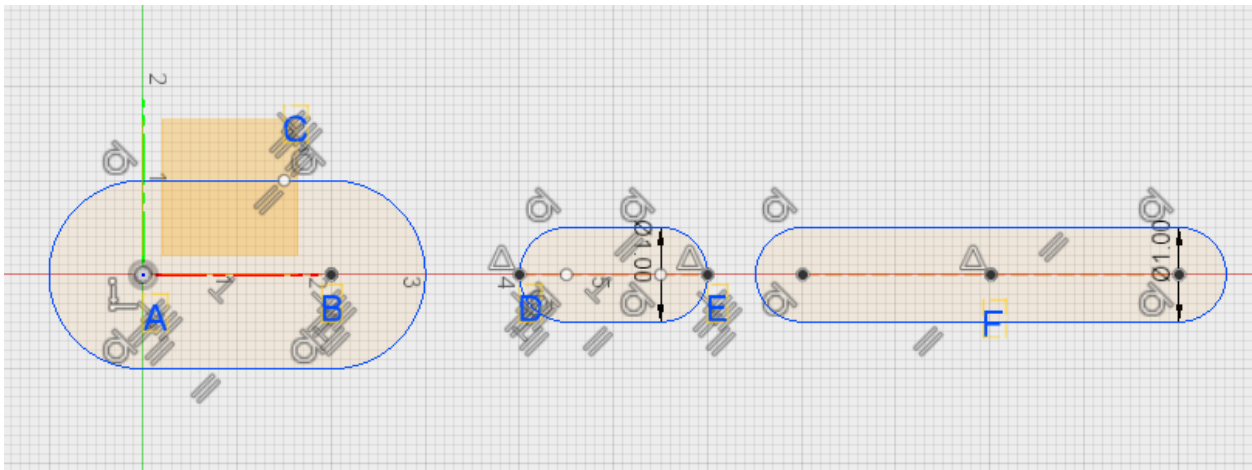
1. Pick one endpoint of the slot
2. Pick the other endpoint of the slot
3. Pick a point on the perimeter or key width

Center Point Slot

1. Pick the center point
2. Pick a point that is the center of one of the circles or key width to the center point of one of the circles
3. Pick a point on the perimeter or key the width of the slot

Try it: Creating Slots

1. Open the MakeICT Fusion 360 Sketching for Makers Project (If not already open)
2. Open the Starter Models folder (If not already open)
3. Open **2. Simple Sketching** (If not already open)
4. Open the Sketches from the browser window (If not already open)
5. Double-Click the Slots sketch to edit it
6. If you cannot see anything in the sketch, pick the lightbulb
7. Sketch>Slot>Center to Center Slot
8. Pick point A
9. Pick point B
10. Pick point C
11. Sketch>Slot>Overall Slot
12. Pick point D
13. Pick point E
14. Key 1
15. Sketch>Slot>Center to Center Slot
16. Pick point F
17. Key 2
18. Key 1
19. Try a few ellipses of your own
20. Select the Stop Sketch button
21. Turn off the sketch (if needed)
22. Save the design



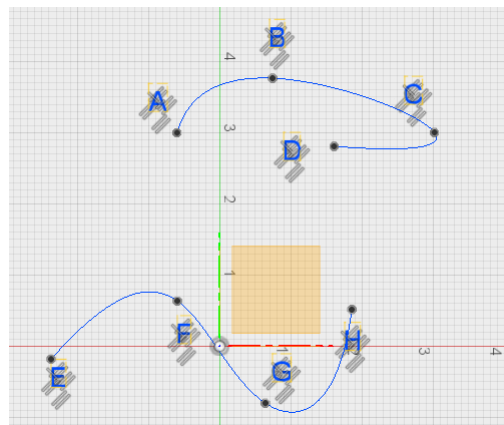
Splines

Splines are curves that are created by picking a set of points for the curve to pass through. They would be good if you were trying to make more artsy elements in a CAD model. (CAD models aren't really that good with artsy things though, so if you want to make something more artsy, you might want to consider a drawing program, like Inkscape.)

1. Select the start point
2. Select points until you are ready to end
3. Double-click the last point or hit the Esc key to end
4. If you have the spline selected, but are not creating a spline, you can move the control points to make the spline change shape.

Try it: Creating Splines

1. Open the MakeICT Fusion 360 Sketching for Makers Project (If not already open)
2. Open the Starter Models folder (If not already open)
3. Open **2. Simple Sketching** (If not already open)
4. Open the Sketches from the browser window (If not already open)
5. Double-Click the Splines sketch to edit it. If you cannot see anything, pick the lightbulb
6. Sketch>Spline
7. Pick point A
8. Pick point B
9. Pick point C
10. Double-click point D to end the spline
11. Pick point E
12. Pick point F
13. Pick point G
14. Pick point H
15. Hit Esc to end the spline
16. Move some of the green points on the spline to see what happens
17. Try a few splines of your own
18. Select the Stop Sketch button
19. Turn off the sketch (if needed)
20. Save the design



Rectangles

Rectangles are one of the most basic shapes. We have three kinds in Fusion 360.

2-Point Rectangle (R)

The 2-point rectangle requires points at diagonal corners. It is the most used of the rectangles and has a shortcut: R

1. Pick one corner point
2. Pick the diagonally opposite corner point or key both the length and width of the rectangle

3-Point Rectangle

The 3-point rectangle is good for rectangle set at an angle.

1. Pick one corner point
2. Pick a second corner point or key the length of the rectangle and select a point in the direction
3. Pick a third corner point or key the width of the rectangle

Center Rectangle

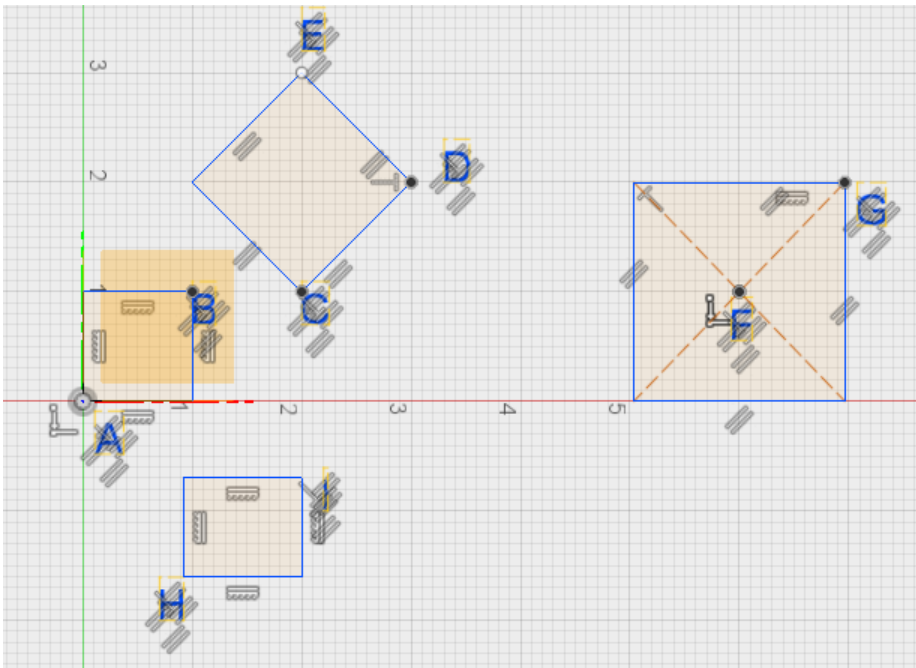
The Center Rectangle requires a center point and a corner point

1. Pick the center point
2. Pick a corner point or key both length and width (which makes it along the axes)

In the following exercise, the last rectangle has you just pick spots near the letters. I put points in for the exercises so that you would have an exact place to pick, but it is not required... you can just let the points nearby and if you have your “snap to grid” turned on, it will make it on an exact gridline or if it is turned off, it will make it in a random location. This is useful if the location is irrelevant or if you will choose to constrain the item later.

Try it: Creating Rectangles

1. Open the MakeICT Fusion 360 Sketching for Makers Project (If not already open)
2. Open the Starter Models folder (If not already open)
3. Open **2. Simple Sketching** (If not already open)
4. Open the Sketches from the browser window (If not already open)
5. Double-Click the Rectangle sketch to edit it. If you cannot see anything, pick the lightbulb
6. Sketch>Rectangle>2-Point Rectangle
7. Pick point A
8. Pick point B
9. Sketch>Rectangle>3-Point Rectangle
10. Pick point C
11. Pick point D
12. Pick point E
13. Sketch>Rectangle>Center Rectangle
14. Pick point F
15. Pick point G
16. Sketch>Rectangle>2-Point Rectangle
17. Pick a spot near H
18. Pick a spot near I
19. Try a few rectangles of your own
20. Select the Stop Sketch button
21. Turn off the sketch (if needed)
22. Save the design



Polygons

Polygons are regularly shaped closed contours of lines. (Triangles, Squares, Pentagons, Hexagons, Heptagons, Octagons, etc.). Inscribed polygons are polygons where the vertexes fall inside a circle; Circumscribed polygons are ones where there is a circle touching the midpoints of the lines making up the polygon.

Circumscribed Polygon

You will want this polygon when you know the distance to one of the edges or have a point at center of one of the edges.

1. Select a center point
2. Key the number of vertices (if needed)
3. Select a center point of an edge or key the distance to the edge

Inscribed Polygon

You will want to use this polygon when you know the distance from the center to one of the vertices,

1. Select the center point
2. Key the number of vertices (if needed)
3. Select a vertex or key the distance to the vertex

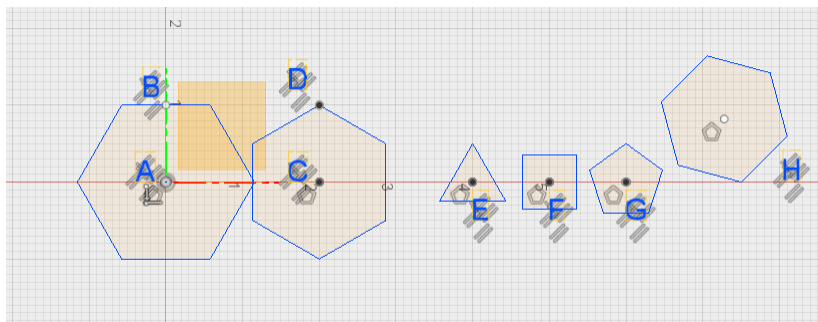
Edge Polygon

An edge polygon takes two vertex points and to determine the orientation and length of the side of the polygon. There are two polygons possible for any two vertices. You pick which one by putting your mouse near the center of the one you want.

1. Select one vertex
2. Select a second adjacent vertex
3. Put cursor on the side of the polygon you want
4. Key the number of vertices (if needed)

Try it: Creating Polygons

1. Open the MakeICT Fusion 360 Sketching for Makers Project (If not already open)
2. Open the Starter Models folder (If not already open)
3. Open **2. Simple Sketching** (If not already open)
4. Open the Sketches from the browser window (If not already open)
5. Double-Click the Polygon sketch to edit it. If you cannot see anything, pick the lightbulb
6. Sketch>Polygon>Circumscribed Polygon
7. Pick point A
8. Key 6 in the number of vertices field
9. Pick point B
10. Sketch>Polygon>Inscribed Polygon
11. Pick point C
12. Key 6 in the number of vertices field (if it is not already there, which we would expect)
13. Pick point D
14. Sketch>Polygon>Inscribed Polygon
15. Pick point E
16. If your cursor is not in the first field, then tab to the next field
17. Key .5 for distance to the vertex
18. Tab to get to the next field
19. Key 3 for number of vertices
20. Sketch>Polygon>Inscribed Polygon
21. Key .5 for distance to the vertex
22. Tab to get to the next field
23. Key 4 for number of vertices
24. Sketch>Polygon>Inscribed Polygon
25. Key .5 for distance to the vertex
26. Tab to get to the next field
27. Key 5 for number of vertices
28. Sketch>Polygon>Inscribed Polygon
29. Pick one of the endpoints of line H
30. Pick the other endpoint of line H
31. Move cursor to the location near the top of the polygon
32. Key 6 in the Vertices field
33. Select the Stop Sketch button
34. Turn off the sketch (if needed)
35. Save the design



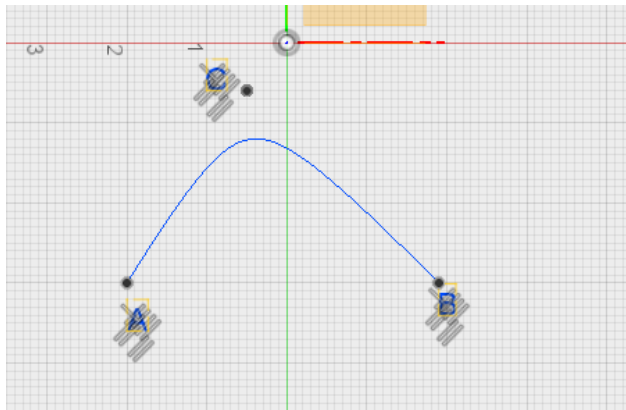
Conics*

Conics are special curves that are very smooth. They are ellipses, hyperbolas and parabolas. These are specialty curves that you are less likely to use.

1. Pick one endpoint
2. Pick a second endpoint
3. Pick the point that is the intersection of the tangents of the endpoints
4. Key the ratio. (75 is a parabola, <75 is an ellipse, >75 is a hyperbola)

Try it: Creating Conics

1. Open the MakeICT Fusion 360 Sketching for Makers Project (If not already open)
2. Open the Starter Models folder (If not already open)
3. Open **2. Simple Sketching** (If not already open)
4. Open the Sketches from the browser window (If not already open)
5. Double-Click the Polygon sketch to edit it. If you cannot see anything, pick the lightbulb
6. Sketch>Polygon>Circumscribed Polygon
7. Pick point A
8. Pick point B
9. Pick point C
10. Key 75 as the parameter (for a parabola)
11. Select the Stop Sketch button
12. Turn off the sketch (if needed)
13. Save the design



Text

The text option creates text in the sketch. You must explode the text if you want to use it as geometry. To create text:

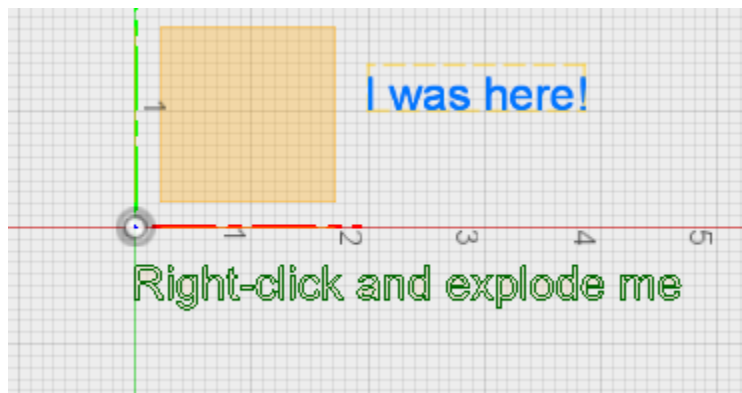
1. Select a location
2. Key text

To explode text:

1. Select a piece of text
2. Right click and pick **Explode Text** from the menu

Try it: Creating Text

- Open the MakeICT Fusion 360 Sketching for Makers Project (If not already open)
- Open the Starter Models folder (If not already open)
- Open **2. Simple Sketching** (If not already open)
- Open the Sketches from the browser window (If not already open)
- Double-Click the Text sketch to edit it. If you cannot see anything, pick the lightbulb
- Sketch>Text
- Select any location
- Key "I was here!"
- Select the "Right-click and explode me" text
- Right-click and pick **Explode Text**
- Try a few more pieces of text on your own
- Select the Stop Sketch button
- Turn off the sketch (if needed)
- Save the design



Geometric Constraints

You might have noticed several blue symbols pop up as you create things. If you make a line that is close to vertical, the program will add a little blue icon that indicates that it will remain vertical and if you try to move the line it might move around on the page, but will remain vertical. You will also have noticed some icons that are little blue parallel lines. As you might expect, this is telling you that a pair of lines that must remain parallel to each other. In the next section, we will show you how to add these kind of constraints, known as geometrical constraints. While you are doing your next exercise, notice the items that geometrically constrain your profiles.

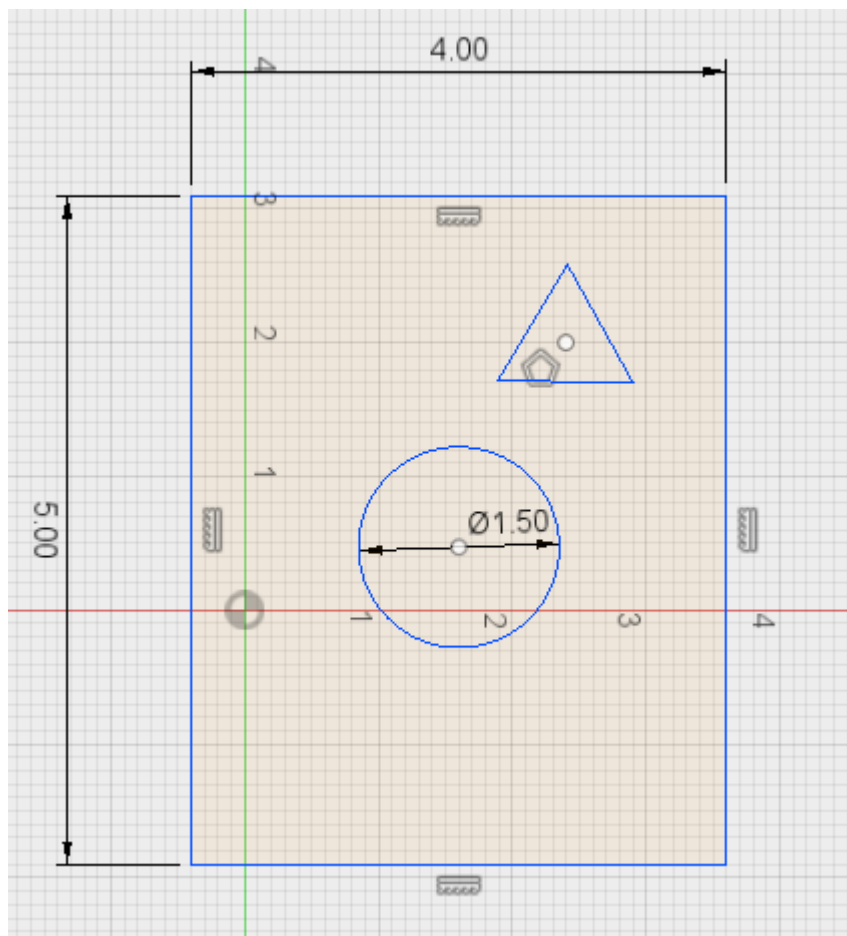
Dimensional Constraints

As we created the simple profiles, we occasionally will put dimensions on lengths, angles, etc. on a part and that will allow us to force the items to those values (and change them to any values we want.) We will also discuss these in the next section; so as you do the next exercise, notice how they work.

Exercise 1

You have watched geometry being created. You have created some geometry following instructions. Now it is time to see how much you have learned.

Create a new design and add the geometry shown in the picture:



Creating Complex Profiles

So far, we have created simple profiles, now we are going to create more complex profiles and learn how to make our own shapes and constrain and modify them.

Profiles

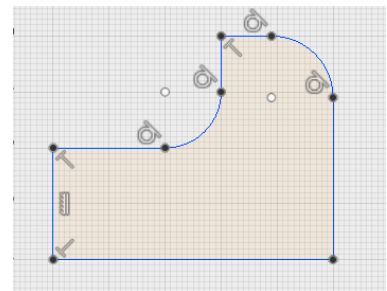
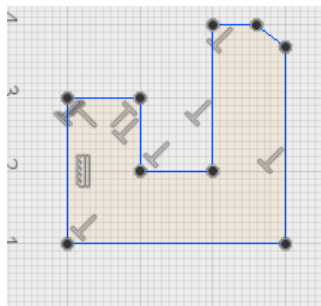
Profiles are a series of curves and lines that are closed (they start and end at the same point.) The easiest way to create a profile is by making lines. You might have noticed that when you create a line, you must explicitly end creating the line. You need to do that because it expects you make more lines and possibly curves. You create a curve by clicking and dragging in the line option. It may take a few minutes to figure out how this works. The curve will be tangent to the previous element.

To make a profile with the line function:

1. Pick a start point
2. Pick some more points and/or click and drag to get a round corner
3. Pick the start point again

Try it: Creating Profiles

1. Open the MakeICT Fusion 360 Sketching for Makers Project (If not already open)
2. Open the Starter Models folder (If not already open)
3. Open **3. Complex Profiles**
4. Open the Sketches from the browser window (If not already open)
5. Double-Click the Profile sketch to edit it. If you cannot see anything, pick the lightbulb
6. Sketch>Line
7. Select one of the points and then the rest of the points to match the first picture below
8. Double click the first point to close the profile and end the line function
9. Stop the sketch
10. Double-Click the **Profile with Corners** sketch to edit it. If you cannot see anything, pick the lightbulb
11. Sketch>Line
12. Select one of the points and then the rest of the points to match the second picture below. You will need to select the point and drag to start the curves. Notice the curves are tangent and always go clockwise.
13. Select the Stop Sketch button
14. Turn off the sketch (if needed)
15. Try a few more profiles on your own
16. Save the design



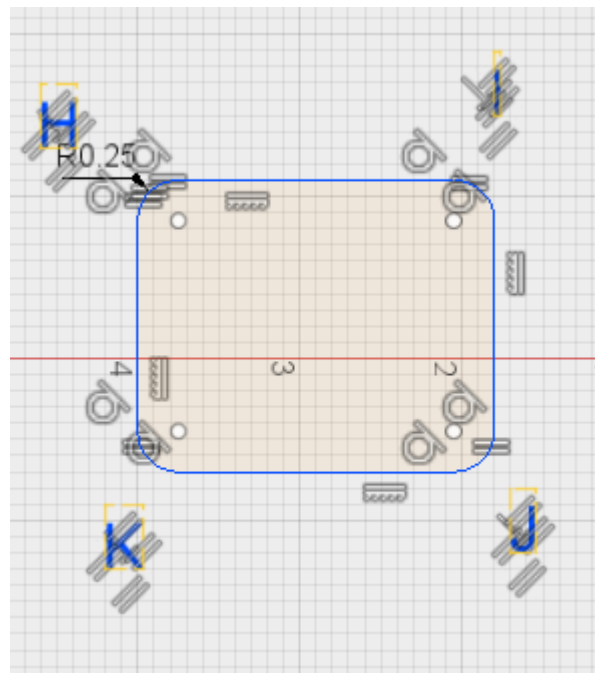
Fillets

Sharp corners on parts could poke you and really don't look as elegant as a smooth corner. Because of this most parts that we make have fillets on them. (Note fillet is pronounced fill-it not fill-lay.)

1. Select an edge or curve
2. Select a second edge or curve
3. Key the radius value

Try it: Creating Fillets

1. Open the MakeICT Fusion 360 Sketching for Makers Project (If not already open)
2. Open the Starter Models folder (If not already open)
3. Open **4. Sketch Operations**
4. Open the Sketches from the browser window (If not already open)
5. Double-Click the Fillet sketch to edit it. If you cannot see anything, pick the lightbulb
6. Sketch>Fillet
7. Select one of the lines near H
8. Select the other line near H
9. Select one of the lines near I
10. Select the other line near I
11. Select one of the lines near J
12. Select the other line near J
13. Select one of the lines near K
14. Select the other line near K
15. Key .25 and press Enter
16. Select the Stop Sketch button
17. Turn off the sketch (if needed)
18. Save the design



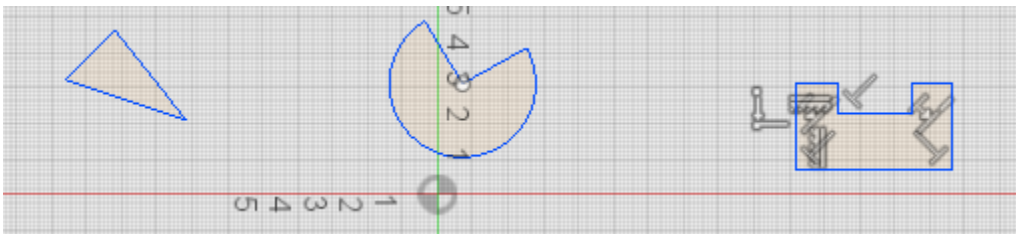
Trim (T)

Trim lets you trim extra bits from curves and lines.

1. Select one line or curve on the piece of the line or curve that should be eliminated
2. Repeat as necessary

Try it: Creating Trim

1. Open the MakeICT Fusion 360 Sketching for Makers Project (If not already open)
2. Open the Starter Models folder (If not already open)
3. Open **4. Sketch Operations** (if needed)
4. Open the Sketches from the browser window (If not already open)
5. Double-Click the Trim sketch to edit it. If you cannot see anything, pick the lightbulb
6. Sketch>Trim
7. Select each of the lines or curves that shouldn't be there to make it look like the picture below
8. Select the Stop Sketch button
9. Turn off the sketch (if needed)
10. Save the design



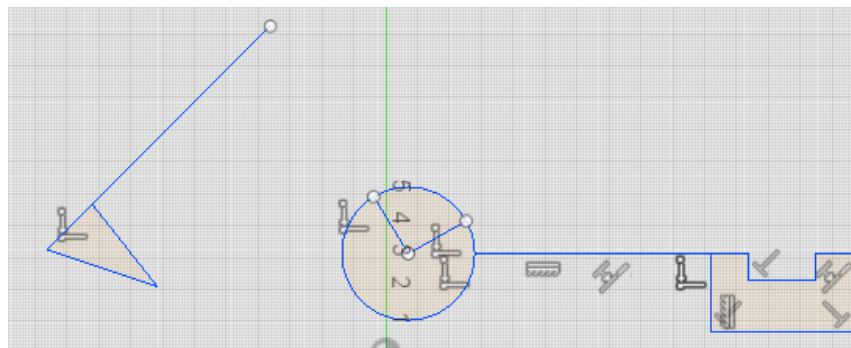
Extend

While trim shortens up elements, extend makes them longer. We will use the trim exercise to extend some of the things we trimmed. (Some things can be extended even if they weren't trimmed.)

1. Hover over the element to see what can be extended
2. Select the element to keep the proposed extension

Try it: Creating Extensions

1. Open the MakeICT Fusion 360 Sketching for Makers Project (If not already open)
2. Open the Starter Models folder (If not already open)
3. Open **4. Sketch Operations** (if needed)
4. Open the Sketches from the browser window (If not already open)
5. Double-Click the Trim sketch to edit it. (If it is not already open). If you cannot see anything, pick the lightbulb
6. Sketch>Extend
7. Hover over the elements
8. Select each of the lines or curves that need to be there to make it look like the picture below
9. Select the Stop Sketch button
10. Turn off the sketch (if needed)
11. Save the design



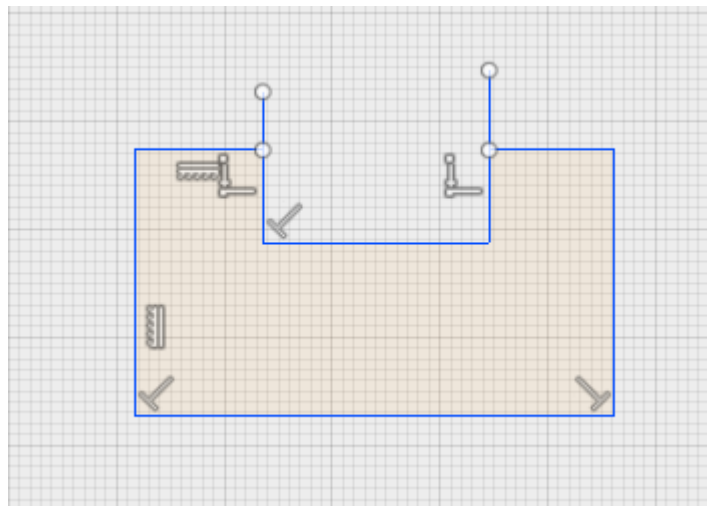
Break

Break will break an element into two elements. Sometimes you will need this to trim.

1. Hover over the element
2. Pick the element to break it

Try it: Creating Breaks

1. Open the MakeICT Fusion 360 Sketching for Makers Project (If not already open)
2. Open the Starter Models folder (If not already open)
3. Open **4. Sketch Operations** (if needed)
4. Open the Sketches from the browser window (If not already open)
5. Double-Click the Break sketch to edit it. If you cannot see anything, pick the lightbulb
6. Sketch>Break
7. Hover over the top line
8. Select the top line to break at the two intersections
9. Select the middle piece of the top line and hit the delete key to see that the line was broken into 3 pieces
10. Select the Stop Sketch button
11. Turn off the sketch (if needed)
12. Save the design



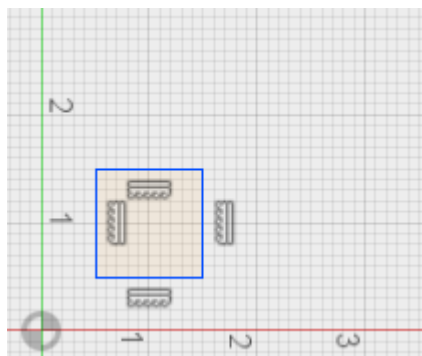
Sketch Scale

Sketch scale will allow you to scale elements in a sketch. This can be useful if you are changing the size of elements in the sketch or if you just made the sketch without regard to size and when adding dimensions, elements like fillets are getting out of shape.

1. Pick all the elements you want to scale. You can box around them, if you like.
2. Select the Point field from the Sketch Scale window that pops up.
3. Select the point you want to scale around
4. Key the ratio you want to scale the sketch in the Scale Factor field

Try it: Creating Sketch Scale

1. Open the MakeICT Fusion 360 Sketching for Makers Project (If not already open)
2. Open the Starter Models folder (If not already open)
3. Open **4. Sketch Operations** (if needed)
4. Open the Sketches from the browser window (If not already open)
5. Double-Click the Scale sketch to edit it. If you cannot see anything, pick the lightbulb
6. Sketch>Sketch Scale
7. Select around all of the lines in the square
8. Select the Point field from Sketch Scale window
9. Select the origin point (0,0)
10. Key .5 into the Scale Factor Field (since we want the part to be half size)
11. Scale a few other things, if you like
12. Turn off the sketch (if needed)
13. Save the design



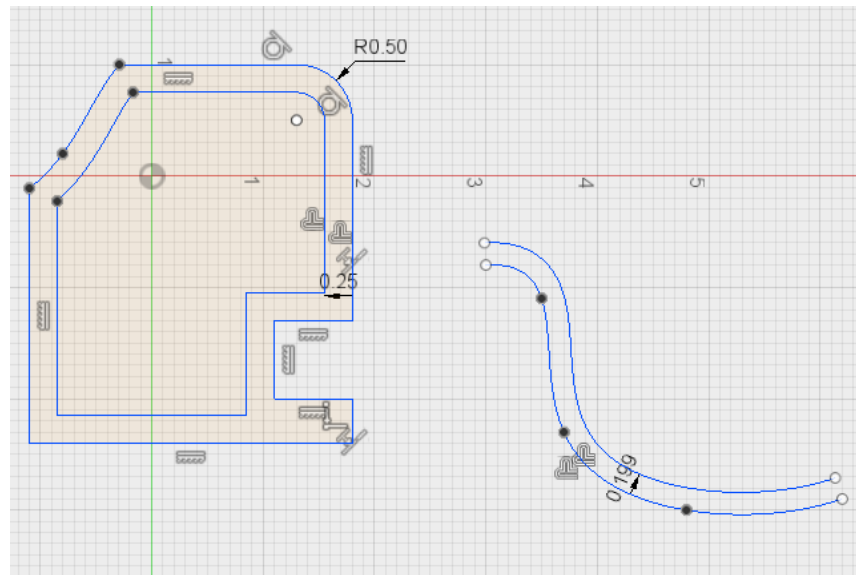
Offset (O)

Offset makes a parallel item that has been scaled to make thicknesses even. So, if you have an exterior contour, the interior contour will be what is needed to keep an even thickness. This is good for modeling things that are evenly thick (like sheet metal) even when they are bent into shape.

1. Select the contour
2. Select the blue <> icon to move the red contour in and or out or key the distance

Try it: Creating an Offset

1. Open the MakeICT Fusion 360 Sketching for Makers Project (If not already open)
2. Open the Starter Models folder (If not already open)
3. Open **4. Sketch Operations** (if needed)
4. Open the Sketches from the browser window (If not already open)
5. Double-Click the Offset sketch to edit it. If you cannot see anything, pick the lightbulb
6. Sketch>Offset
7. Select any element of the contour
8. Pull the blue <> icon both inside and outside the contour so you can see what options you have for offset
9. Key .25 in the offset field
10. Pick the curve
11. Use the blue <> icon to pull it out to a thickness that you like and press enter
12. Offset a few other things, if you like
13. Turn off the sketch (if needed)
14. Save the design



Mirror

If a part is symmetric, mirror will let you create one half of the part and get the second half without having to do all the drawing.

First create a construction line to mirror about

1. Create a line where you want to mirror about
2. Make the line be a construction line

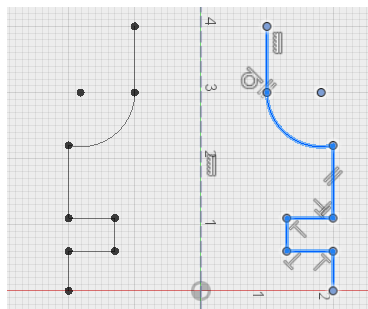
Mirror the geometry

1. Select all the geometry to mirror
2. Select the construction line

Try it: Creating Mirrored Geometry

1. Open the MakeICT Fusion 360 Sketching for Makers Project (If not already open)
2. Open the Starter Models folder (If not already open)
3. Open **4. Sketch Operations** (if needed)
4. Open the Sketches from the browser window (If not already open)
5. Double-Click the Mirror sketch to edit it. If you cannot see anything, pick the lightbulb
6. Sketch>Mirror
7. Select all the lines on the right
8. Select the Mirror Line field in the Mirror window that pops up
9. Select the center line (that dotted line up the middle that we made construction for you)
10. Select ok from the Mirror window
11. Change a few lines to be construction lines by right clicking and picking construction/normal
12. Mirror a few other things, if you like
13. Turn off the sketch (if needed)
14. Save the design

You may have noticed that I did not make lines across the top and bottom. I did this on purpose so I could create them as a single line. (Often the line being in two parts could cause issues down the line.)



Circular Pattern

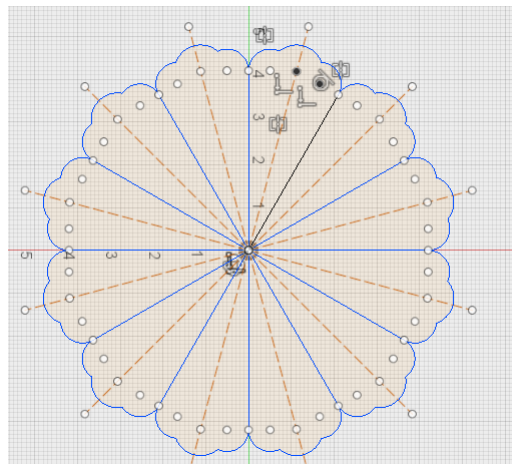
Circular pattern will let you copy elements into a circular pattern. This will make for spectacular geometric designs.

1. Select the elements that make up the pattern (Make sure and not select the center point)
2. In the Pattern window, pick the Point field
3. Pick the center point
4. Choose the type Full, Angle or Symmetric (full is a full circle, angle is a certain number of degrees beginning with the selected elements and going clockwise, Symmetric is also a given number of degrees, but symmetric about the geometry given)
5. Key the angle (only for Angle or Symmetric)
6. Key the number of rotations elements you would like to see in the pattern

Try it: Creating a Circular Pattern

1. Open the MakeICT Fusion 360 Sketching for Makers Project (If not already open)
2. Open the Starter Models folder (If not already open)
3. Open **4. Sketch Operations** (if needed)
4. Open the Sketches from the browser window (If not already open)
5. Double-Click the Circular Pattern sketch to edit it. If you cannot see anything, pick the lightbulb
6. Sketch>Circular Pattern
7. Select all the contour except the center point (should be 15 elements.... Ctrl+Select the center point to remove it if you have 16)
8. Select in the point field and pick the center point
9. Make sure the Type field is set to full
10. Make sure suppress is checked (so you can remove some copies if you wish)
11. Key 12 in the Quantity field and press Enter or select OK
12. Pattern a few other things, if you like
13. Turn off the sketch (if needed)
14. Save the design

Notice I created a construction line here to symmetry part of the geometry around.



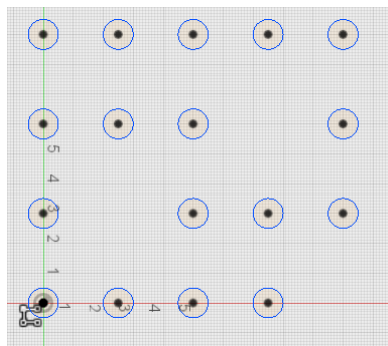
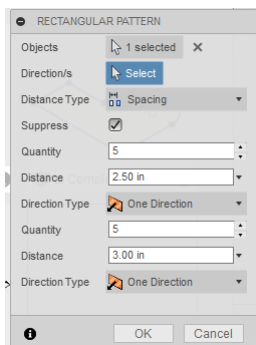
Rectangular Pattern

Rectangular pattern gives you copies of items along two directions.

1. Select the geometry that you want to pattern
2. Select the Direction field and select a line or lines direction you want to pattern in (so the lines don't have to be perpendicular; it could be a diamond shaped pattern.)
3. Select if you want to specify the total length (extent) or the spacing between items
4. Check the suppress box if you want to be able to turn off some of the items
5. Choose the quantity for the first direction and key in how many items you want in a row
6. Choose the distance (total or spacing depending on what you picked in item 3.)
7. Choose the direction type one direction or symmetric
8. Choose the quantity for the second direction and key in how many items you want in a row
9. Choose the distance (total or spacing depending on what you picked in item 3.)
10. Choose the direction type one direction or symmetric

Try it: Creating a Rectangular Pattern

1. Open the MakeICT Fusion 360 Sketching for Makers Project (If not already open)
2. Open the Starter Models folder (If not already open)
3. Open **4. Sketch Operations** (if needed)
4. Open the Sketches from the browser window (If not already open)
5. Double-Click the Rectangular Pattern sketch to edit it. If you cannot see anything, pick the lightbulb
6. Sketch>Rectangular Pattern
7. Select the circle
8. Select in the point field and pick the center point of the circle
9. Make sure the distance type is set to Spacing
10. Make sure suppress is checked (so you can remove some copies if you wish)
11. Key 5 in the first Quantity field
12. Key 2.5 in the first Distance field
13. Key 6 in the second Quantity field
14. Key 3.0 in second distance field
15. Uncheck a couple of the circles to suppress them
16. Press the OK button
17. Pattern a few other things, if you like
18. Turn off the sketch (if needed)
19. Save the design



Constraints & Construction

Constraints are certain rules that we can force geometry to adhere to. There are a number of physical properties we can constrain (like tangency, concentricity, parallelism)

We may have to construct some geometry that is just intermediate geometry to our final product. This is known as construction geometry.

Construction Geometry

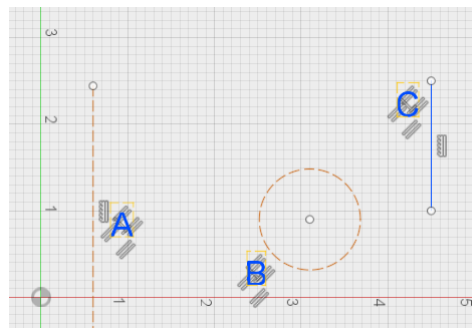
As you have seen, if we need to make symmetric elements, we may have to create construction elements in preparation. Creating construction elements is easy. We just create the element we want and turn it into a construction type.

1. Make a piece of geometry (like a line)
2. Select the piece of geometry (if needed)
3. Right-click and select Construction/Normal

Note if you need to turn a piece of construction geometry back to normal, you follow the same steps as it is a toggle switch.


Try it: Construction Geometry

1. Open the MakeICT Fusion 360 Sketching for Makers Project (If not already open)
2. Open the Starter Models folder (If not already open)
3. Open **5. Construction and Constraints** (if needed)
4. Open the Sketches from the browser window (If not already open)
5. Double-Click the Constraints sketch to edit it. If you cannot see anything, pick the lightbulb
6. Select the line near A
7. Right-click to get the Marking Menu and select the Construction/Normal option
8. Select the circle near B
9. Right-click to get the Marking Menu and select the Construction/Normal option
10. Select the construction circle near C
11. Right-click to get the Marking Menu and select the Construction/Normal option
12. Change construction or normal to a few other things, if you like
13. Turn off the sketch (if needed)
14. Save the design





Coincident

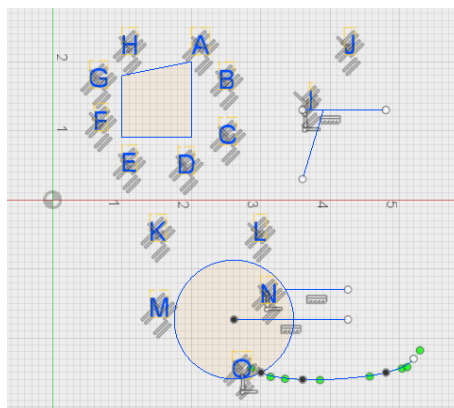
One of the easiest constraints to understand is coincident. Coincident points mean the two points have the same coordinates. A point that is coincident with a line means the point is on the line. A point that is coincident with a circle means that the point is on the circle.

Select the coincident icon from the Sketch Palette 

1. Select a geometrical element
2. Select a second geometrical element


Try it: Coincident

1. Open the MakeICT Fusion 360 Sketching for Makers Project (If not already open)
2. Open the Starter Models folder (If not already open)
3. Open **5. Construction and Constraints** (if needed)
4. Open the Sketches from the browser window (If not already open)
5. Double-Click the Coincident sketch to edit it. If you cannot see anything, pick the lightbulb
6. Select the coincident icon from the Sketch Palette 
7. Select the point near A
8. Select the point near B
9. Use coincident to make the other corners of the square meet (CD,EF,GH). Note: the order the points are chosen will determine which point moves... so, it might not exactly match the picture below.
10. Select the point on the top of the I line
11. Select the J line toward the middle. This attaches that point to the line.
12. Select the line near L
13. Select the circle near K and the circle will be on the endpoint of the line
14. Select the center point of the circle (near M)
15. Select the endpoint of the line near N and the line will attach to the center of the circle
16. Select the spline near O and the circle anywhere on the perimeter to make the endpoint of the line stick to the circle
17. Click the  icon again to turn it off
18. Grab a few points or other pieces of geometry and see how it reacts
19. Turn off the sketch (if needed)
20. Save the design





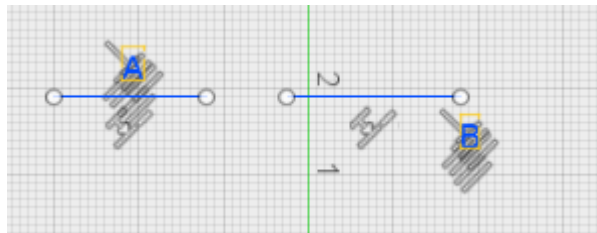
Collinear

Lines that are collinear have the same orientation, but not the same length or endpoints

1. Pick the collinear icon 
2. Pick a line
3. Pick another line


Try it: Collinear

1. Open the MakeICT Fusion 360 Sketching for Makers Project (If not already open)
2. Open the Starter Models folder (If not already open)
3. Open **5. Construction and Constraints** (if needed)
4. Open the Sketches from the browser window (If not already open)
5. Double-Click the Collinear sketch to edit it. If you cannot see anything, pick the lightbulb
6. Select the collinear icon 
7. Select Line A
8. Select Line B and the line will align with A
9. Click the  icon again to turn it off
10. Move the lines around to see what happens
11. Turn off the sketch (if needed)
12. Save the design





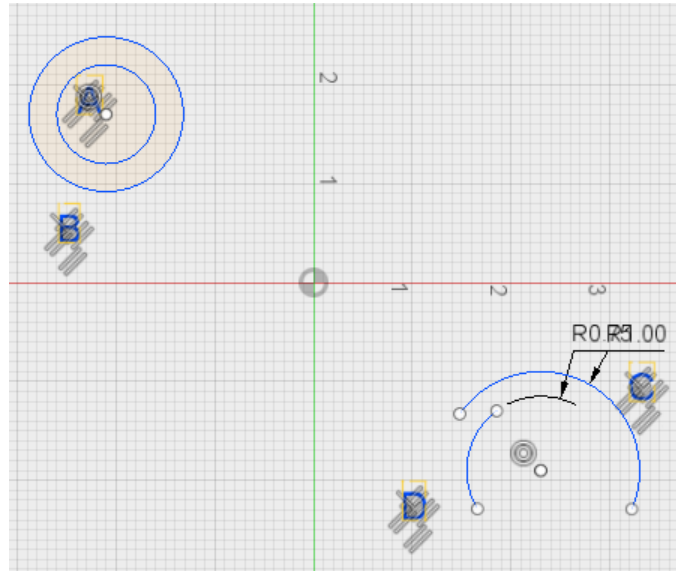
Concentric

Concentric makes circles have the same center.

1. Pick the concentric icon 
2. Pick a circle or arc
3. Pick another circle or arc


Try it: Concentric

1. Open the MakeICT Fusion 360 Sketching for Makers Project (If not already open)
2. Open the Starter Models folder (If not already open)
3. Open **5. Construction and Constraints** (if needed)
4. Open the Sketches from the browser window (If not already open)
5. Double-Click the Concentric sketch to edit it. If you cannot see anything, pick the lightbulb
6. Select the concentric icon from the Sketch Palette 
7. Select Circle A
8. Select Circle B
9. Select arc C
10. Select Arc D
11. Click the  icon again to turn it off
12. Move the lines around to see what happens
13. Turn off the sketch (if needed)
14. Save the design





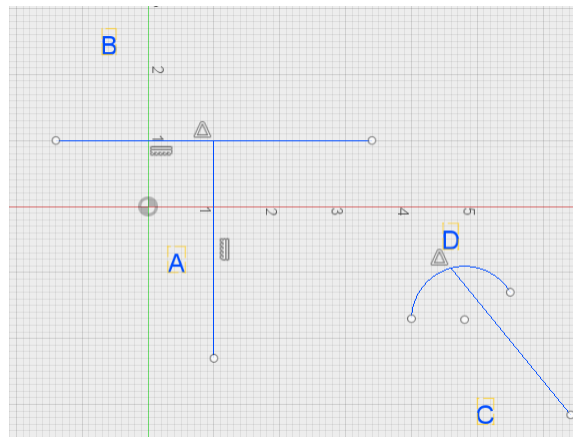
Midpoint

Midpoint attaches a point onto a midpoint of another element.

1. Pick the Midpoint icon 
2. Pick the point
3. Pick the element to attach that point to


Try it: Midpoint

1. Open the MakeICT Fusion 360 Sketching for Makers Project (If not already open)
2. Open the Starter Models folder (If not already open)
3. Open **5. Construction and Constraints** (if needed)
4. Open the Sketches from the browser window (If not already open)
5. Double-Click the Midpoint sketch to edit it. If you cannot see anything, pick the lightbulb
6. Select the midpoint icon from the Sketch Palette 
7. Select the top endpoint of line A
8. Select line B
9. Select the top endpoint of line D
10. Select circle C
11. Click the  icon again to turn it off
12. Move the lines and circles around to see what happens
13. Turn off the sketch (if needed)
14. Save the design





Fix/Unfix

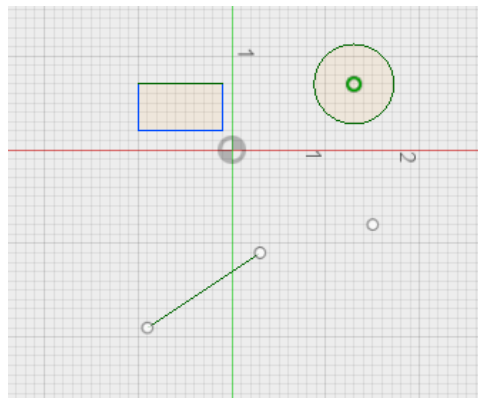
When a piece of geometry is fixed, it will not move at all (as though it were glued down)

1. Pick the Fix/Unfix icon 
2. Pick the piece of geometry

Try it: Fix/Unfix


1. Open the MakeICT Fusion 360 Sketching for Makers Project (If not already open)
2. Open the Starter Models folder (If not already open)
3. Open **5. Construction and Constraints** (if needed)
4. Open the Sketches from the browser window (If not already open)
5. Double-Click the Fix sketch to edit it. If you cannot see anything, pick the lightbulb
6. Select the Fix icon from the Sketch Palette 
7. Select the top line on the square
8. Select the circle
9. Select the line
10. Click the  icon again to turn it off
11. Move the lines and circles around to see what happens
12. Turn off the sketch (if needed)
13. Save the design

Notice that all the lines that are fixed have turned a dark green. That is a clue that an element is fully constrained (you won't be able to drag those parts at all. Also notice the icon is named fix/unfix. If you need to remove the fix from a part, you will do the same steps to toggle it off.





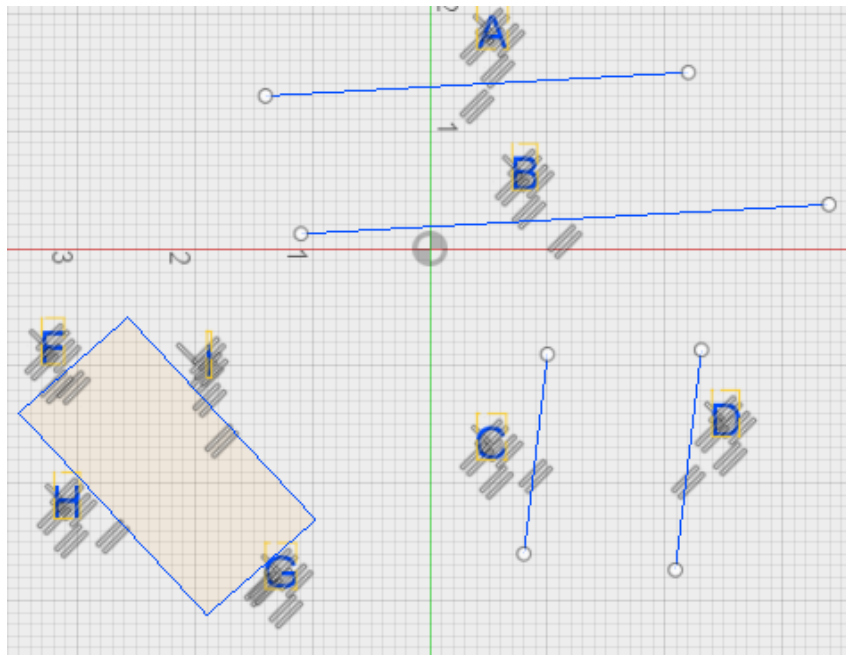
Parallel

Parallel forces one line to be parallel to another one.

1. Pick the Parallel icon 
2. Pick a line
3. Pick a second line to be parallel to the first


Try it: Parallel

1. Open the MakeICT Fusion 360 Sketching for Makers Project (If not already open)
2. Open the Starter Models folder (If not already open)
3. Open **5. Construction and Constraints** (if needed)
4. Open the Sketches from the browser window (If not already open)
5. Double-Click the Parallel sketch to edit it. If you cannot see anything, pick the lightbulb
6. Select the Parallel icon from the Sketch Palette 
7. Pick line A
8. Pick line B
9. Pick line C
10. Pick line D
11. Pick line F
12. Pick line G
13. Pick line H
14. Pick line I
15. Click the  icon again to turn it off
16. Move the lines around to see what happens
17. Turn off the sketch (if needed)
18. Save the design





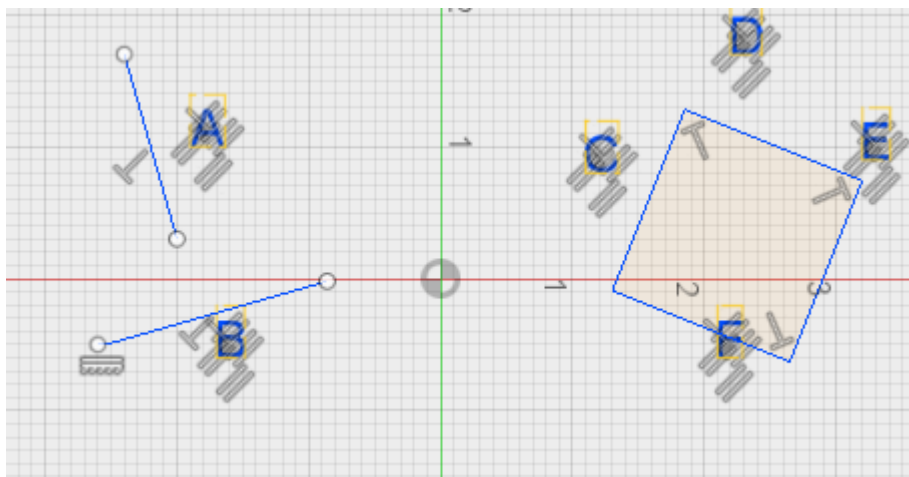
Perpendicular

Perpendicular forces a line to be perpendicular to another line.

1. Pick the Perpendicular icon 
2. Pick one line
3. Pick a second line to be perpendicular to the first


Try it: Perpendicular

1. Open the MakeICT Fusion 360 Sketching for Makers Project (If not already open)
2. Open the Starter Models folder (If not already open)
3. Open **5. Construction and Constraints** (if needed)
4. Open the Sketches from the browser window (If not already open)
5. Double-Click the Perpendicular sketch to edit it. If you cannot see anything, pick the lightbulb
6. Select the Perpendicular icon from the Sketch Palette 
7. Pick line A
8. Pick line B
9. Pick line C
10. Pick line D
11. Pick line D
12. Pick line E
13. Pick line E
14. Pick line F
15. Click the  icon again to turn it off
16. Move the lines around to see what happens
17. Turn off the sketch (if needed)
18. Save the design





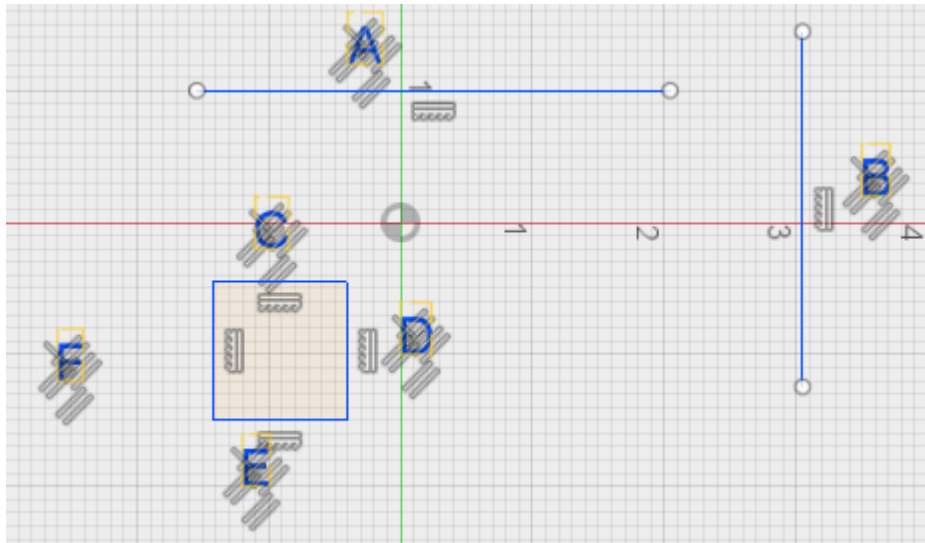
Horizontal/Vertical

The Horizontal/Vertical icon will force a line to be either horizontal or vertical (whichever is closer when you add the constraint)

1. Pick the Horizontal/Vertical icon 
2. Pick the line you want to make either horizontal or vertical


Try it: Horizontal/Vertical

1. Open the MakeICT Fusion 360 Sketching for Makers Project (If not already open)
2. Open the Starter Models folder (If not already open)
3. Open **5. Construction and Constraints** (if needed)
4. Open the Sketches from the browser window (If not already open)
5. Double-Click the Horizontal/Vertical sketch to edit it. If you cannot see anything, pick the lightbulb
6. Select the Horizontal/Vertical icon from the Sketch Palette 
7. Pick line A
8. Pick line B
9. Pick line C
10. Pick line D
11. Pick line E
12. Pick line F
13. Click the  icon again to turn it off
14. Move the lines around to see what happens
15. Turn off the sketch (if needed)
16. Save the design





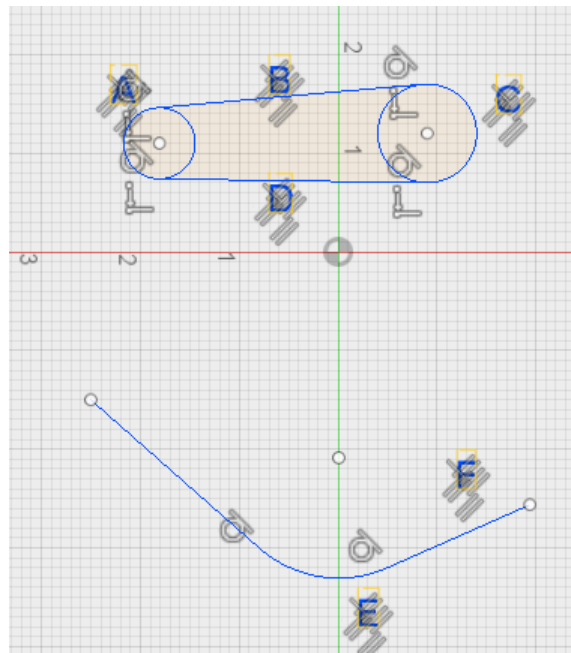
Tangent

Tangent makes the transitions between a line and a curve or between curves be “not pointy”.

1. Pick the tangent icon 
2. Pick one of the elements to be tangent
3. Pick another element to be tangent


Try it: Tangent

1. Open the MakeICT Fusion 360 Sketching for Makers Project (If not already open)
2. Open the Starter Models folder (If not already open)
3. Open **5. Construction and Constraints** (if needed)
4. Open the Sketches from the browser window (If not already open)
5. Double-Click the Tangent sketch to edit it. If you cannot see anything, pick the lightbulb
6. Select the Tangent icon from the Sketch Palette 
7. Pick circle A
8. Pick line B
9. Pick line B
10. Pick circle C
11. Pick circle C
12. Pick line D
13. Pick line D
14. Pick circle A
15. Pick circle E
16. Pick line F
17. Click the  icon again to turn it off
18. Move the lines and circles around to see what happens
19. Turn off the sketch (if needed)
20. Save the design





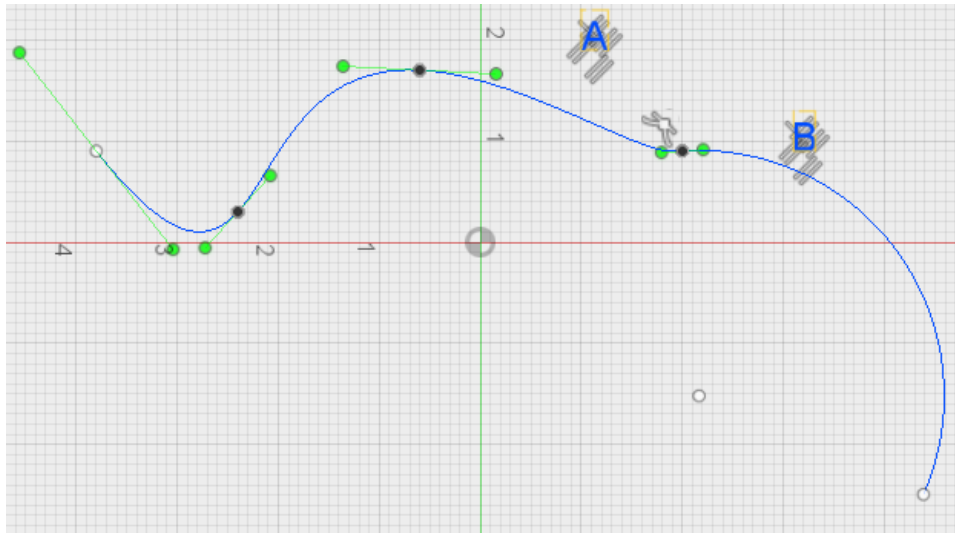
Smooth*

Smooth makes a smooth transition between curves. It makes both the tangency and curvature the same at the transition.

1. Pick the smooth icon 
2. Pick one curve
3. Pick a second curve


Try it: Smooth

1. Open the MakeICT Fusion 360 Sketching for Makers Project (If not already open)
2. Open the Starter Models folder (If not already open)
3. Open **5. Construction and Constraints** (if needed)
4. Open the Sketches from the browser window (If not already open)
5. Double-Click the Smooth sketch to edit it. If you cannot see anything, pick the lightbulb
6. Select the Smooth icon from the Sketch Palette 
7. Pick curve A
8. Pick curve B
9. Click the  icon again to turn it off
10. Move the curves around to see what happens
11. Turn off the sketch (if needed)
12. Save the design





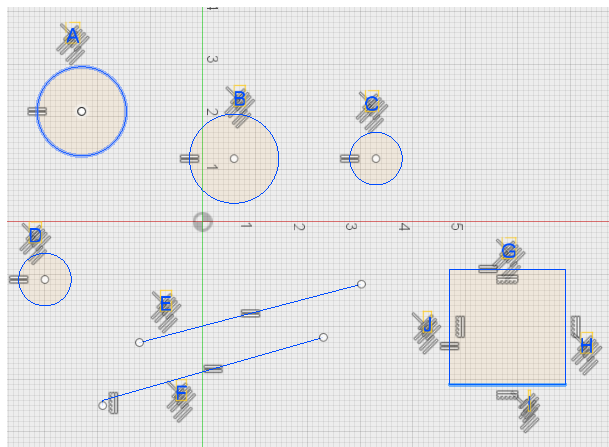
Equal

Equal makes two elements have equal dimensions. So lines will be the same length and circles will be the same radius. It does not change the element's location.

1. Select the equal icon 
2. Select the reference element
3. Select the element you want to match the first element

Try it: Equal

1. Open the MakeICT Fusion 360 Sketching for Makers Project (If not already open)
2. Open the Starter Models folder (If not already open)
3. Open **5. Construction and Constraints** (if needed)
4. Open the Sketches from the browser window (If not already open)
5. Double-Click the Equal sketch to edit it. If you cannot see anything, pick the lightbulb
6. Select the Equal icon from the Sketch Palette 
7. Pick circle A
8. Pick circle B
9. Pick circle C
10. Pick circle D
11. Pick line E
12. Pick line F
13. Pick line G
14. Pick line H
15. Pick line G
16. Pick line I
17. Pick line G
18. Pick line J
19. Click the  icon again to turn it off
20. Move the lines and circles around to see what happens
21. Turn off the sketch (if needed)
22. Save the design



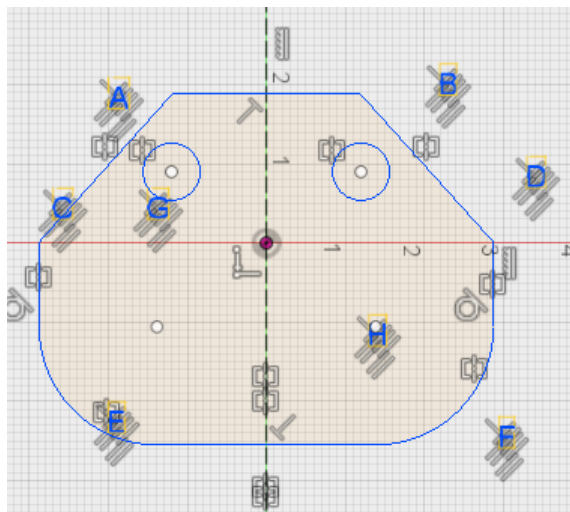
Symmetry

Symmetry is like mirror (except you must have the geometry already existing)

1. Select the Symmetry icon
2. Select a piece of geometry
3. Select a piece of geometry to be symmetric
4. Select the centerline (If one doesn't exist, you may have to construct this line and make it a construction line)

Try it: Symmetry

1. Open the MakeICT Fusion 360 Sketching for Makers Project (If not already open)
2. Open the Starter Models folder (If not already open)
3. Open **5. Construction and Constraints** (if needed)
4. Open the Sketches from the browser window (If not already open)
5. Double-Click the Symmetry sketch to edit it. If you cannot see anything, pick the lightbulb
6. Select the Symmetry icon from the Sketch Palette =
7. Pick line A
8. Pick line B
9. Pick centerline
10. Pick line C
11. Pick line D
12. Pick centerline
13. Pick curve E
14. Pick curve F
15. Pick centerline
16. Pick circle G
17. Pick circle H
18. Pick centerline
19. Click the = icon again to turn it off
20. Move the lines and circles around to see what happens
21. Turn off the sketch (if needed)
22. Save the design



Dimension (D)

Dimensional constraints will constrain things like lengths, angles, radius and diameter.

Lengths

1. Sketch>Dimension
2. Pick an element like a line
3. Pick a location to set the dimension
4. Change the dimension or hit enter

Between two elements

1. Sketch>Dimension
2. Pick an element (like a line or point)
3. Pick a second element (like a line or point)
4. Pick a location for the dimension
5. Change the dimension or hit enter

Diameter

1. Sketch>Dimension
2. Pick a circle
3. Pick a location for the dimension
4. Change the dimension or hit enter


Radius

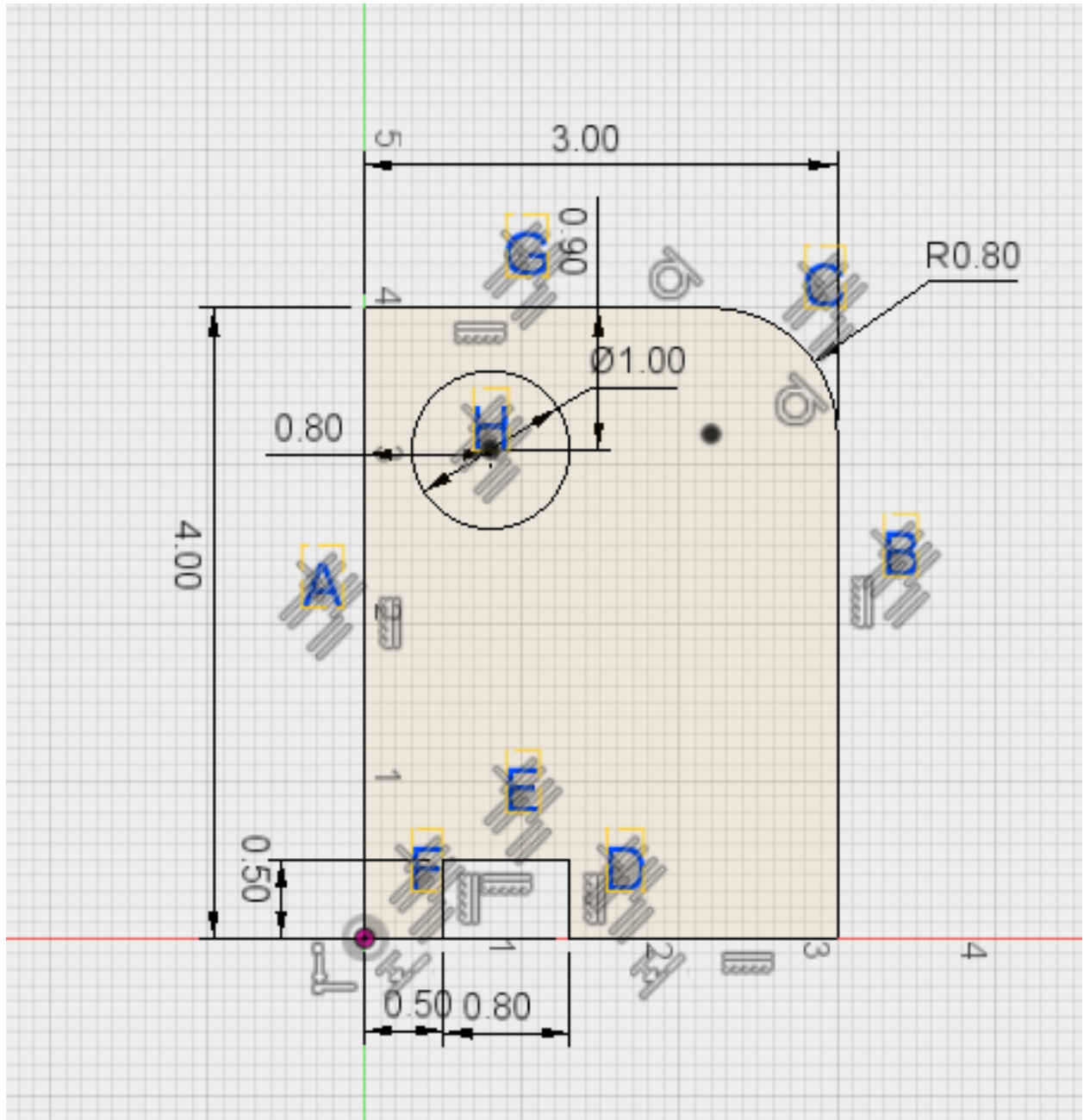
1. Sketch>Dimension
2. Pick an arc or circle
3. Pick a location for the dimension
4. Change the dimension or hit enter
5. Right click the dimension and choose Toggle Radius (if needed)

Angle

1. Sketch>Dimension
2. Pick a line
3. Pick a second line
4. Move the dimension around to get the correct quadrant
5. Change the dimension or hit enter

Try it: Dimension

1. Open the MakeICT Fusion 360 Sketching for Makers Project (If not already open)
2. Open the Starter Models folder (If not already open)
3. Open **5. Construction and Constraints** (if needed)
4. Open the Sketches from the browser window (If not already open)
5. Double-Click the Symmetry sketch to edit it. If you cannot see anything, pick the lightbulb
6. Select the Symmetry icon from the Sketch Palette 
7. Pick line A
8. Pick a location for the dimension
9. Press enter to confirm value
10. Pick line A
11. Pick line B
12. Pick a location for the dimension
13. Press enter to confirm value
14. Pick fillet C
15. Pick a location for the dimension
16. Press enter to confirm value
17. Pick circle H
18. Pick a location for the dimension
19. Change the value to 1.00
20. Press enter to confirm value
21. Pick the center point of circle H
22. Pick line A
23. Pick a location for the dimension
24. Press enter to confirm value
25. Pick the center point of circle H
26. Pick line B
27. Pick a location for the dimension
28. Press enter to confirm value
29. Pick line F
30. Pick line A
31. Pick a location for the dimension
32. Press enter to confirm value
33. Pick line F
34. Pick line D
35. Pick a location for the dimension
36. Press enter to confirm value
37. Move the lines and circles around to see what happens
38. The only thing that will move is that line E will be able to move up and down... this is how you check to see if you are constrained
39. Pick line E
40. Pick either of the two bottom lines
41. Pick a location for the dimension
42. Press enter to confirm value
43. Try a few dimensions on your own
44. Turn off the sketch (if needed)
45. Save the design



Importing and Exporting

Importing

To import an SVG or a DXF:

1. Insert
2. SVG or DXF (whichever you have)
3. Navigate to your file
4. Pick open and the file imports into your current sketch

Try it: Importing

1. Open the MakeICT Fusion 360 Sketching for Makers Project (If not already open)
2. Open the Starter Models folder (If not already open)
3. Open **7. Importing and Exporting**
4. Open the Sketches from the browser window (If not already open)
5. Right-click over the sketch named Import here and choose Insert SVG
6. Navigate to your desktop and find the file called H1.svg
7. Press the save button

Exporting

To export a sketch:

1. Right-click over the sketch
2. Save as DXF

NOTE: The DXF that Fusion exports is a newer version and will not work in the laser cutter. You will need to import it into a software (like Inkscape) that will convert it to DXF R14

Try it: Exporting

1. Open the MakeICT Fusion 360 Sketching for Makers Project (If not already open)
2. Open the Starter Models folder (If not already open)
3. Open **7. Importing and Exporting**
4. Open the Sketches from the browser window (If not already open)
5. Right-click over the sketch named Export Me and choose Save as DXF
6. Navigate to your desktop
7. Rename the model to Export.dxf
8. Press the save button

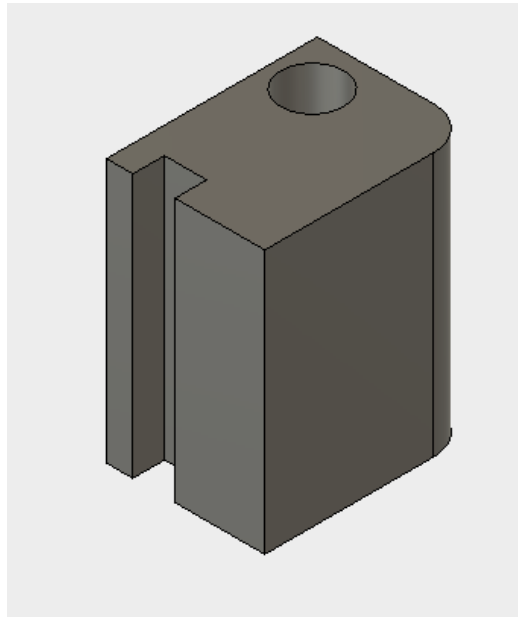
Making Extrudes

A little bonus preview of solids. We will make an extrude from one of our sketches.

1. Open the model containing your sketch
2. Create>Extrude (E)
3. Select your sketch
4. Key distance of extrusion

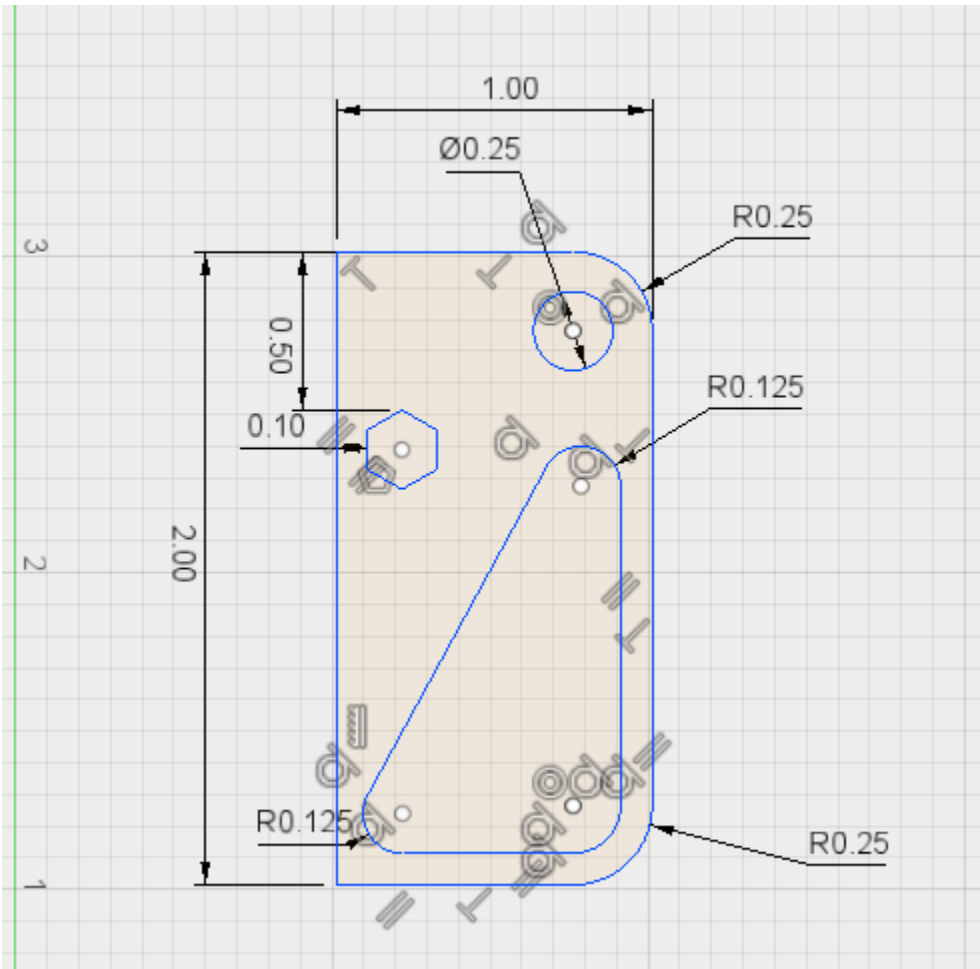
Try it: Extrude

1. Open the MakeICT Fusion 360 Sketching for Makers Project (If not already open)
2. Open the Starter Models folder (If not already open)
3. Open **8. Extrudes**
4. Open the Sketches from the browser window (If not already open)
5. Model>Extrude
6. Select the sketch
7. Key 5 for the distance
8. Press the save button



Exercise 2

1. Create the sketch shown
2. Create a solid from the sketch that is 2.5 thick
3. Modify the sketch so that the top dimension is 1.25 (and notice what happens to the solid)
4. Export the changed sketch as a DXF
5. Save the solid



Final Projects

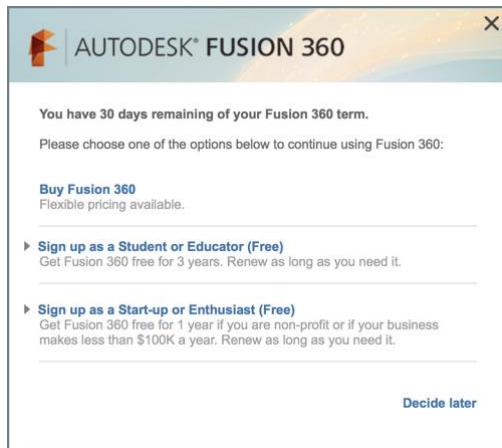
The best final project is one that you really want to use. So, if you have something you want to make, that is your final project.

If you can't think of something to make, get with your instructor for a project

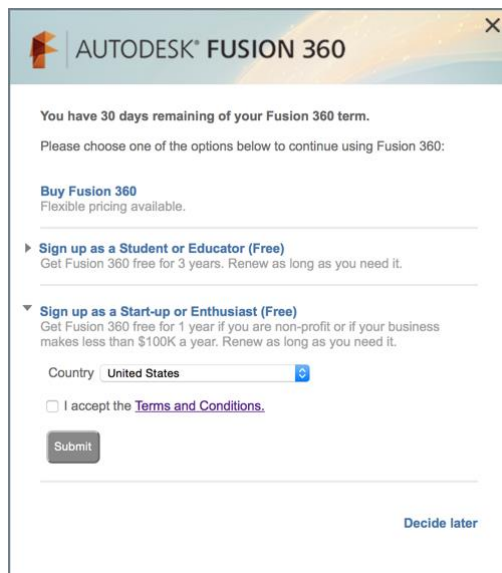
Appendix A: How to download Fusion 360 for your computer

We assume that you have a windows computer for this. To run Fusion 360 on an apple computer is more complex as you would have to download Vine first to allow it to run so we will limit our instructions to downloading for a Microsoft computer.

1. Download Fusion 360 from the [Autodesk](#) website.
2. Select Trial Counter in the upper toolbar of Fusion 360.
3. Select Register for Free Use
4. Select Sign up as a Start-Up or Enthusiast (Free)



5. Read the Terms and Conditions
6. Check the box in front of Terms and Conditions and select Submit



Appendix B: Keyboard Shortcuts and Navigation

Mouse Navigation		
Pan	Hold Middle Mouse Button	Hold Middle Mouse Button
Zoom	Roll Middle Mouse Button	Roll Middle Mouse Button
Orbit	Hold Shift + Hold Middle Mouse Button	Hold Shift + Hold Middle Mouse Button
Orbit around point	Hold Shift + Click then Hold Middle Mouse Button	Hold Shift + Click then Hold Middle Mouse Button

Command	PC	Mac
Undo	Ctrl + Z	Command + Z
Redo	Ctrl + Y	Command + Y
Copy	Ctrl + C	Command + C
Paste	Ctrl + V	Command + V
Cut	Ctrl + X	Command + X

Geometric Element or Operation	Shortcut
Line	L
Center Diameter Circle	C
2-Point Rectangle	R
Trim	T
Dimension	D
Offset	O