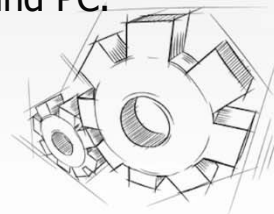# Fusion 360 for FTC

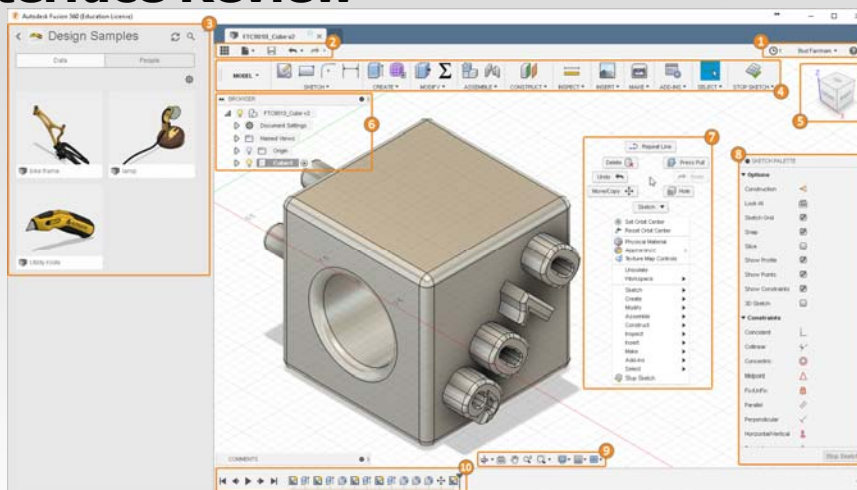## Overview and Highlights

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What is Fusion 360?

- Fusion 360 is a cloud-based CAD/CAM tool for collaborative product development. The tools in Fusion enable exploration and iteration on product ideas and collaboration within a product development team. Fusion 360 combines fast and easy organic modeling with precise solid modeling, allowing you to make your designs manufacturable.

- It connects your entire product development process in a single cloud-based platform that works on both Mac and PC.

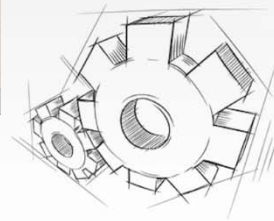- This is great for FIRST Tech Challenge Teams!

Interface Review

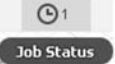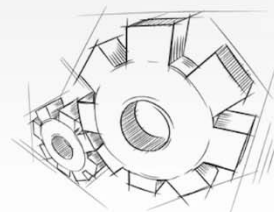# Interface Review

Job Status / Profile / Help

Job Status - Lets the user know if Fusion 360 is online or offline. It also tracks cloud-based jobs such as imports and simulations.

User Profile - Access to the user's account information, personal preferences and the Fusion 360 website with all the user's drawings. This menu also offers the ability to logout of Fusion 360.

Help - Contains access to online learning content, help, forums, step-by-step tutorials, or link to community content.

Interface Review

Application Bar

②

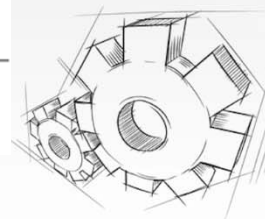|	
⊞	**Data Panel** - Display or hide the data panel. Within the data panel is all the projects. The projects are used for data management and collaboration.
📄 ▾	**File** - Create a New Design, Drawing, Save, Save As, Export, and 3D Print the current model.
💾	**Save** - Save an untitled design with a name or save the current changes to an existing design
↩ ▾ ↪ ▾	**Undo/redo** - Undo/redo operations.

Interface Review

③ **Data Panel** - The Data Panel provides a new way of working with models, saving them, uploading your own files, uploading files from manufacturers and cooperating with other people.
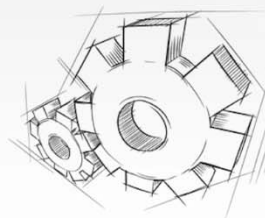
Interface Review

(4)

Workspaces - The workspaces within Fusion 360 are purpose/task driven. Changing the workspace changes relevant commands and tools, and feedback to reinforce actions are available.

Model – Creates mechanical designs that contain mostly prismatic geometry. Access commands to create solid bodies.

Patch – Creates construction surfaces and repairs surface geometry. Access commands to create surface bodies.

Sheet Metal – Creates sheet metals designs. Use these commands to set rules, create sheet metal bodies and product manufacturing data.

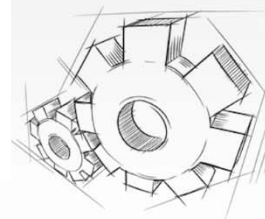**Render** – Generates realistic renderings of the design.

Animation – Creates animations of how the design should be operated or assembled.

CAM – Generates tool-paths and G-code for the design for subtractive manufacturing.

Drawing – Generates 2 manufacturing drawings.

Interface Review

(5)

ViewCube – Use the ViewCube to orbit or view the design or view from standard view positions.
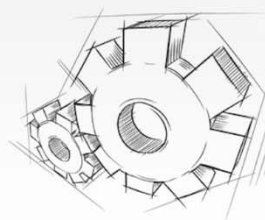
(6)

Browser – The browser lists objects in your design. Use the browser to make changes to objects and control visibility of objects.

(7)

Marking Menu – Right-click to access the marking menu. The marking menu contains frequently used commands in the wheel and all commands in the overflow menu.

Interface Review

8	**Sketch Constraints** - Constraints relate sketch entities and points to one another geometrically.
9	**Navigation bar and display settings** - The navigation bar contains commands used to zoom, pan, and orbit your design. The display settings control the appearance of the interface and how designs are displayed in canvas.
10	**Timeline** - The timeline lists operations performed on your design. Right-click operations in the timeline to make changes. Drag operations to change the order they are calculated.
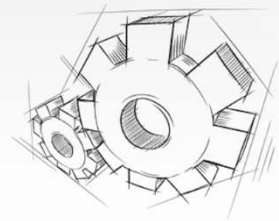
Working with Projects

- Fusion 360 organizes and manages data using a centralized, cloud-based, collaborative platform. This enables robotics teams to work easily and efficiently together. This powerful and secure set of tools dramatically improves the way robotics teams can design, visualize, simulate, and share information.

- The main way Fusion 360 controls access is by using "projects." Projects are unique locations where teams keep all related information in one shared place. Teams can share and access design data, discuss challenges and successes, and stay current with project activities.
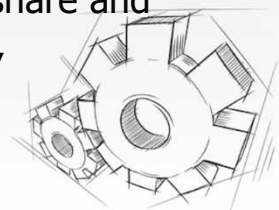
Projects: Lesson 1 – Create a Project

Step 1 – Display the "Data" panel
1. If the "Data Panel" is not already visible, click the "Show Data Panel" button.
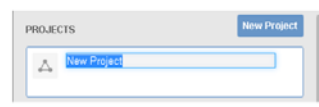2. This button is a toggle on/off.

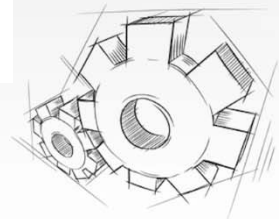**Step 2** – Create a new project
1. Click the "New Project" button

Step 3 – Type the name of your new project
1. After typing the project name, left-click out of the box and the new project is created.
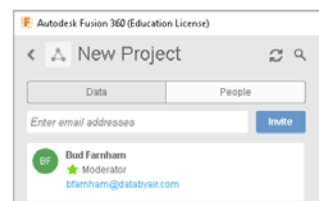
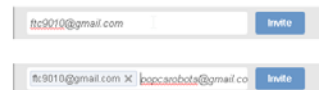FTC 9010 – Robotic Rangers

Projects: Lesson 2 – Add People to Project

Step 1 – Show the "People" in the project
1. Click the "People" button.

Step 2 – Invite people to the project
1. Type an email address where it states "Enter email address"
2. Type a semi-colon (;) after each address if want to enter multiple email addresses.
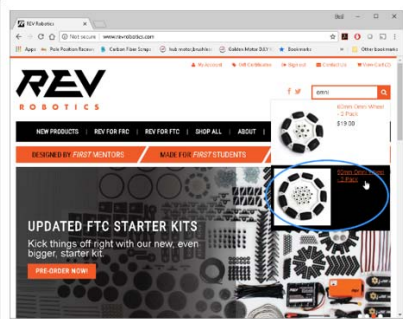3. Click the "Invite" button
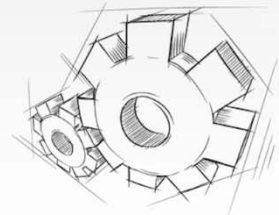
FTC 9010 – Robotic Rangers
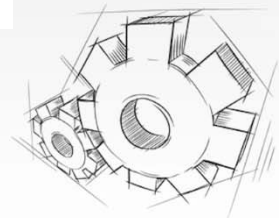
Projects: Lesson 3 – Upload a part

Step 1 – Navigate to REV's website
1. Navigate to http://www.revrobotics.com/
2. In the search box, type "omni"
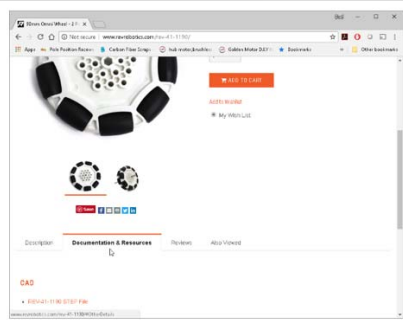3. Click the "90MM Omni Wheel" which is returned in the search results.
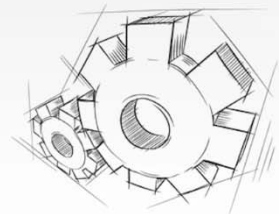
FTC 9010 – Robotic Rangers

Projects: Lesson 3 – Upload a part

Step 2 – Find the STEP file
1. Scroll down the page to the bottom
2. Click the "Documentation & Resources" tab.
3. Notice the "CAD" displays a link to the "REV-41-1190 STEP File".

FTC 9010 – Robotic Rangers

Projects: Lesson 3 – Upload a part

Step 3 – Upload the file to Fusion 360
1. Click the "Upload" button

Step 4 – Navigate to the downloaded file
1. Either click the "Select Files" to open a file-explorer window or drag the file to the "Drag and Drop Here" area.

FTC 9010 – Robotic Rangers
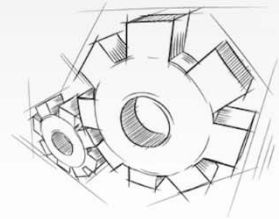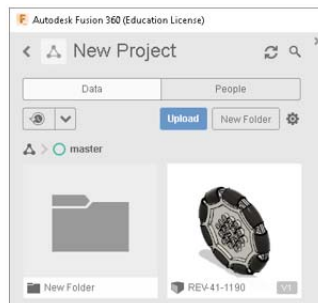

Projects: Lesson 3 – Upload a part

Step 5 – Start the upload
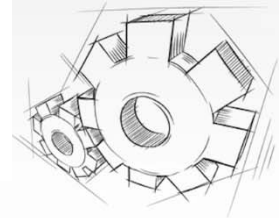1. Press the "Upload" button to begin the upload process.

Step 6 – File upload complete
1. Once complete, the "Refresh" button may need to be pressed to display the newly uploaded part.

FTC 9010 – Robotic Rangers

Projects: Lesson 4 – Open a project

Step 1 – Open the AndyMark TileRunner Chassis
1. Right-click to see the various options.
2. Click "Open" to edit the design.
3. Double-clicking the left mouse button will also open the design.
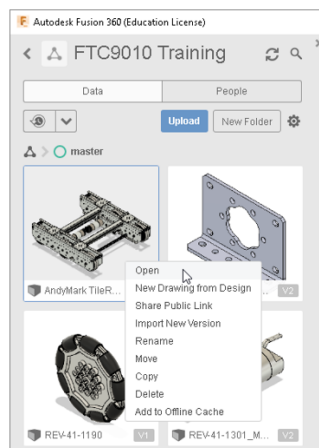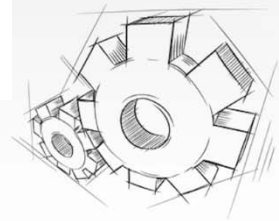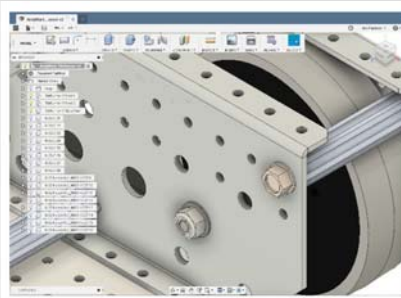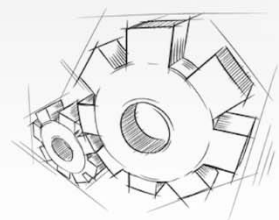
Navigation: Lesson 1 – Zoom

Step 1 – Learning to Zoom In and Zoom Out
1. Roll the scroll-wheel forward or backwards to zoom in and out.
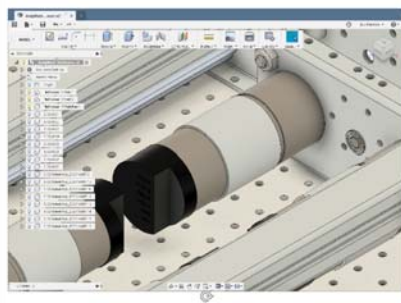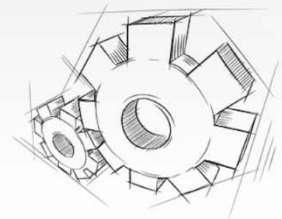2. Notice the zoom in and out is based on the position of the mouse cursor.

Navigation: Lesson 2 – Pan

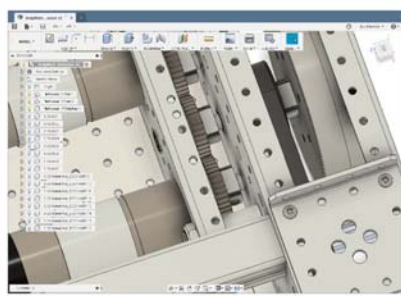**Step 1** – Learning to Pan
1. While holding the scroll-wheel down, move the mouse around.
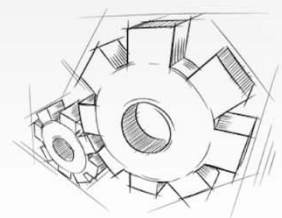
Navigation: Lesson 3 – Orbit

Step 1 – Learning to Orbit
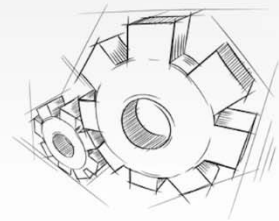1. While holding shift-key and the scroll-wheel down, move the mouse around.

Sketching

- Creating a sketch in Fusion 360 is the basic building block of a design. In order to create intelligent and predictable designs, a good understanding of how to create sketches and how to apply dimensions and geometric constraints is needed. We will start simple and get more complex on making some robot parts for FTC.
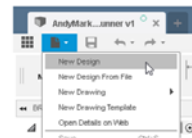
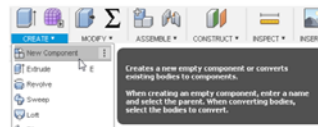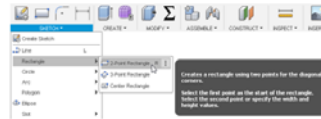# Sketch: Lesson 1 – Draw a cube

Sketch: Lesson 1 – Draw a cube

Step 4 – Create a rectangle
1. There are a couple of methods to create the rectangle.
2. Under "Sketch" in the model workspace, highlight "Rectangle", then "2-Point Rectangle".
3. Notice the "R" in the menu. This is a the shortcut key to a "2-Point Rectangle"
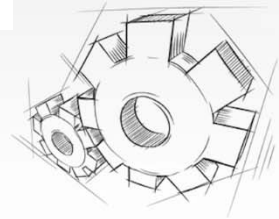
Step 5 – Selecting the sketch plane
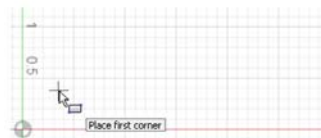1. Click the "Z-plane" which is the top plane.
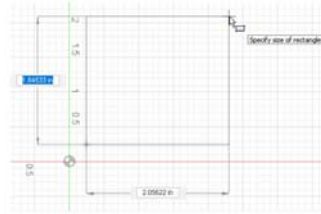

Sketch: Lesson 1 – Draw a cube

Step 6 – Place the first corner of the rectangle
1. Move the cursor to a space anywhere in the plane.
2. Do not select the center of the drawing.
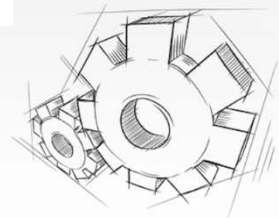3. Left-click once to place the first corner.

Step 7 – Set the second corner of the rectangle.
1. Move the cursor to the top-right.
2. Notice the values are changing as the cursor moves.
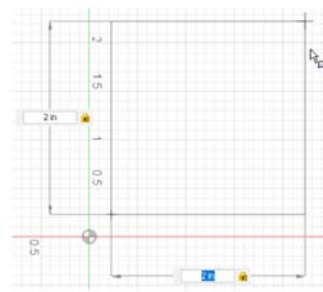3. Notice the blue-highlighted number on the left.
4. Type 2

Sketch: Lesson 1 – Draw a cube

Step 8 – Set the second corner of the rectangle.
1. Press the "Tab" key
2. Notice the blue-highlighted number changes to the number at the bottom of the rectangle.
3. Type 2 again.
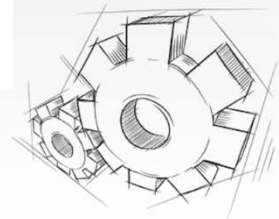4. Press "Enter" on the keyboard
5. The rectangle is now complete.

Step 9 – Complete the sketch
1. Press "Stop Sketch" in the Sketch Palette.

Sketch: Lesson 1 – Draw a cube

Step 10 – Start to Extrude the rectangle
1. Orbit the newly created rectangle so it is not flat.
2. Click "Create" in the Model workspace.
3. Select "Extrude" in the menu.
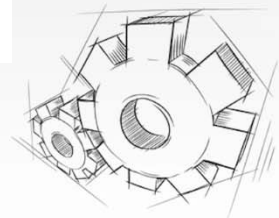4. Notice the shortcut key of "E" could be pressed for "Extrude"

Step 11 – Select the plane to Extrude
1. Move the cursor over the rectangle.
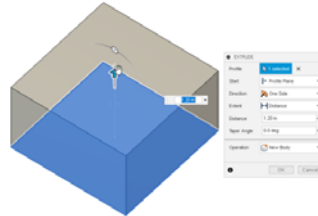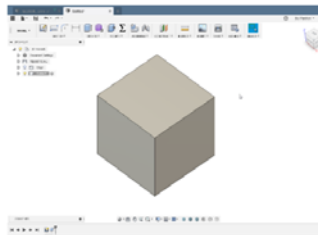2. Left-click with the mouse to select.

Sketch: Lesson 1 – Draw a cube

Step 12 – Set the height of the cube.
1. Grab the arrow and move it up and down.
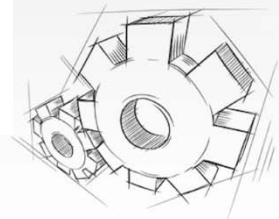2. Notice the value changing as the cursor moves.
3. Type 2
4. Press "OK"

Step 13 – Cube complete!
1. Use the Zoom, Pan, and Orbit commands to move the cube around.
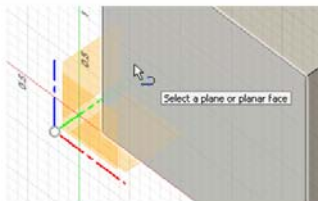

Sketch: Lesson 2 – Make a hole in the cube

Step 1 – Put a hole in the cube
1. Under "Sketch" in the model workspace, highlight and select "Line"
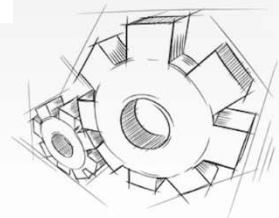2. Notice the "L" in the menu. Pressing "L" is the shortcut key to a "Line"

Step 2 – Select the plane to work with
1. Click the "Front" plane
2. The design will rotate to the plane
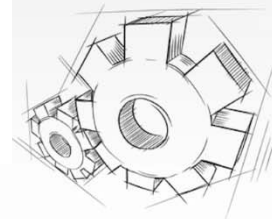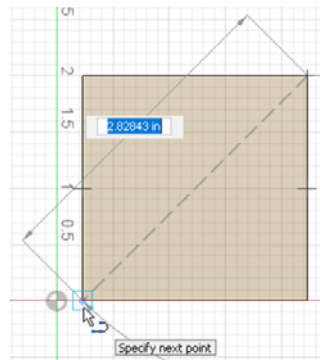
Sketch: Lesson 2 – Make a hole in the cube

Step 3 – Draw construction line
1. Before starting to draw the line, enable the "Construction" mode in the Sketch palette.
2. The shortcut toggle key is "X"

Step 4 – Left-click one corner of the cube.
1. Notice the selection will "snap" to the corner of the cube.
2. Move across the cube diagonally.
3. Left-click at the second corner on the cube.
4. Notice the new line is dashed. This is a construction line and <u>can not</u> be extruded.

2.82843 in

Specify next point

FTC 9010 – Robotic Rangers
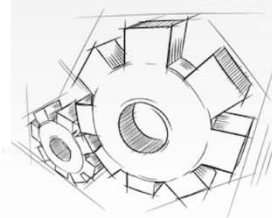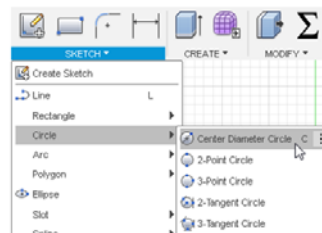

Sketch: Lesson 2 – Make a hole in the cube

Step 5 – Disable construction mode
1. Click the "Construction" mode in the Sketch palette to disable it.
2. The construction shortcut toggle key is "X"

Step 6 – Put a hole in the cube
1. Under "Sketch" in the model workspace, highlight and select "Circle"
2. Notice the "C" in the menu. Pressing "C" is the shortcut key to a "Circle"

SKETCH ▼   CREATE ▼   MODIFY ▼

- Create Sketch
- Line   L
- Rectangle
- Circle   ▶   Center Diameter Circle C
- Arc   2-Point Circle
- Polygon   3-Point Circle
- Ellipse   2-Tangent Circle
- Slot   3-Tangent Circle
- Spline

FTC 9010 – Robotic Rangers
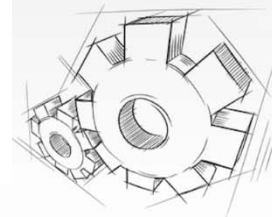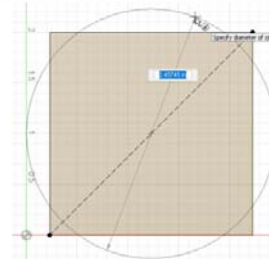
Sketch: Lesson 2 – Make a hole in the cube

Step 7 – Place the circle
1. Hover over the recently created line.
2. Notice the cursor changes. This is due to a constraint.
3. Move towards the middle of the line. Notice the triangle which appears. This indicates the middle of the line.
4. Click once the triangle is showing to place the circle at the middle of the line.
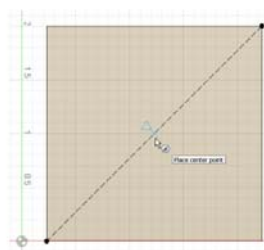
Step 8 – Finish circle creation
1. Do not position the mouse near another line, edge, or point.
2. Click anywhere outside the square to specify the diameter of the circle.

FTC 9010 – Robotic Rangers
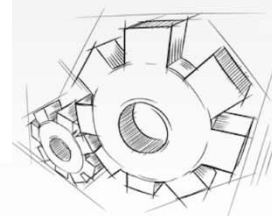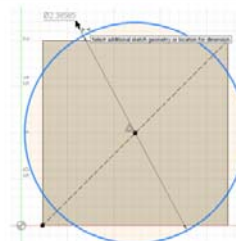

Sketch: Lesson 2 – Make a hole in the cube

Step 9 – Specify the circle diameter dimension
1. Under "Sketch" in the model workspace, select "Dimension"
2. Notice the "D" in the menu. Pressing "D" is the shortcut key to a "Dimension"
3. Place the cursor on the recently created circle.
4. Left-click on the circle.
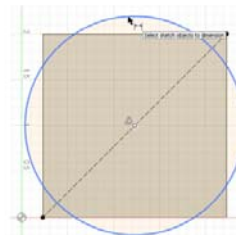
Step 10 – Start to dimension the new circle.
1. Navigate to where the dimension will be placed.
2. Left-click when the dimension is located where desired.
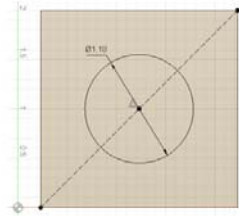3. Type 1.1 and press enter.

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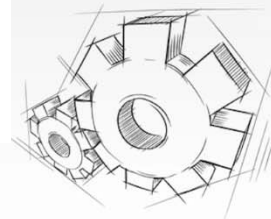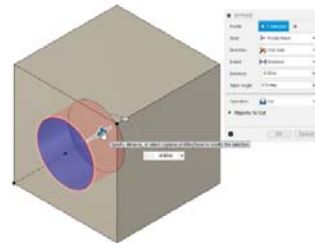# Sketch: Lesson 2 – Make a hole in the cube

Step 11 – Complete dimensioning the circle.
1. Type 1.1 and press enter.
2. The new circle is now defined, and the circle is fully constrained.

Step 12 – Extrude the circle through the cube.
1. Press "Stop Sketch"
2. Orbit the cube to any position
3. Press "E" to extrude.
4. Start to drag the arrow inwards to the cube.
5. Notice the "Operation" will automatically change from "Join" to "Cut" depending on the direction.
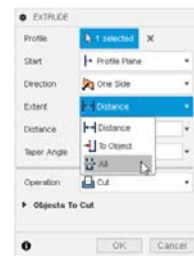

# Sketch: Lesson 2 – Make a hole in the cube

Step 13 – Specify the extrusion length.
1. Open the "Extent" drop-down.
2. The default is "Distance".
3. We could type the cube dimension manually, but if the size of the cube changes, the extrusion dimension would need to change.
4. Select "All" from the drop-down.
5. This will always make a hole through the part.

Step 14 – Hole is complete through the cube.
1. Use the orbit tool to move the cube around.
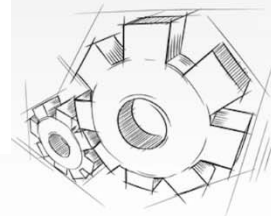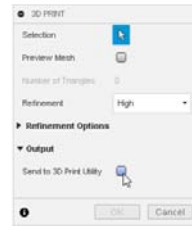
Sketch: Lesson 3 – 3D Print the Cube

Step 1 – Select the "3D Print" command
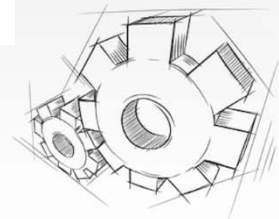1. Under the "Make" icon in the model workspace, select "3D Print"

Step 2 – Uncheck "Send to 3D Print Utility"
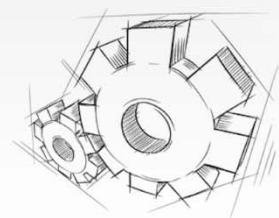1. This box is checked by default.
2. Uncheck the box

Sketch: Lesson 3 – 3D Print the Cube

Step 3 – Select the cube
1. Click anywhere on the cube
2. Press "OK" and a file dialogue box will appear asking where to save the 3D print version of the file (.stl).
3. Press "Save" and save the file.
4. Open the file in your favorite slicer program. Some 3D printers can accept the .stl file directly.
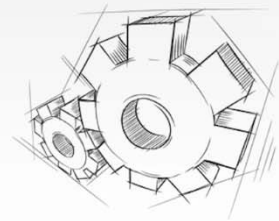
Inspection

- Fusion 360 offers some powerful tools to help check key measurements for either design or comparison with the real world. These item are located under the "Inspect" menu.
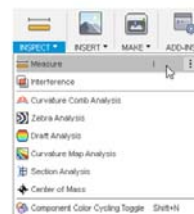

Inspect: Lesson 1 – Measure robot width

Step 1 – Start the measure command
1. Under "Inspect" in the model workspace, highlight and select "Measure"
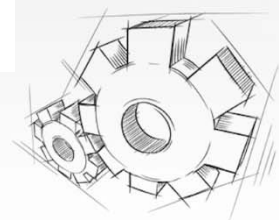2. Notice the "I" in the menu. Pressing "I" is the shortcut key to "Measure" or "Inspect"

Step 2 – Select the first position to measure from
1. Left-click the vertical line on the front-left part of the chassis.
2. A point at the top or bottom could be selected also, but then the same point would need to be selected on the other side.
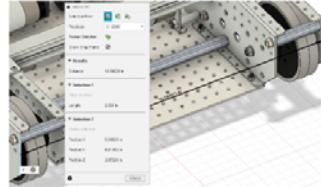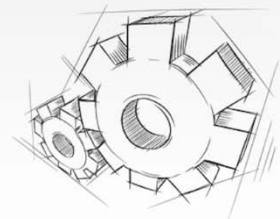
Inspect: Lesson 1 – Measure robot width

Step 3 – Select the second position to measure to
1. Left-click the vertical line on the front-right part of the chassis.
2. The box changes displaying the resulting distance. This distance is 16.569 inches.
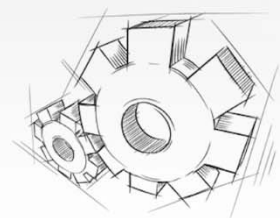
Assemble

- Joints control the position and motion between components.
- Joints indicate the allowable motion between components.

Questions?

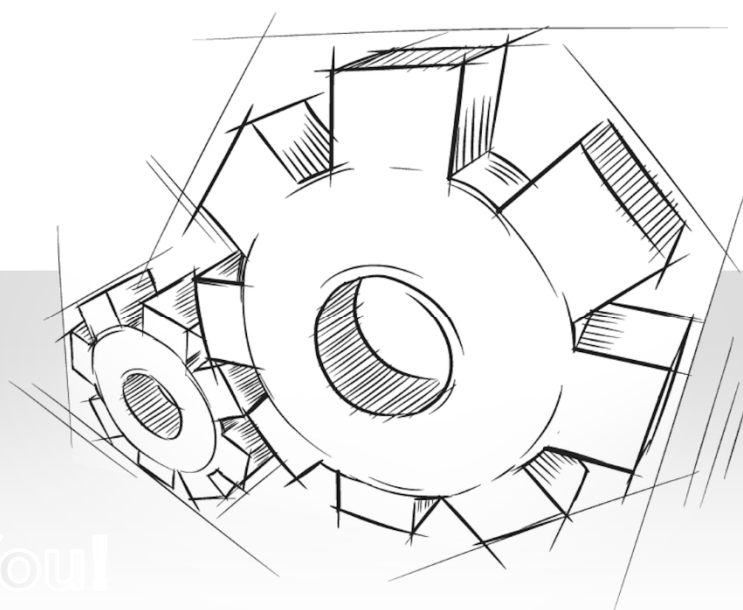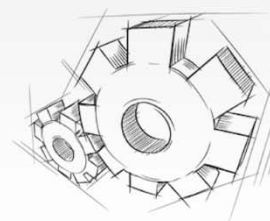- For additional help, contact FTC9010@databyair.com

FTC 9010 – Robotic Rangers

Thank You!