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Level up from Tinkercad to Fusion 360: Design a helmet practice project

Step-by-step guide

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This step-by-step guide is designed to accompany the **Level up From Tinkercad to Fusion 360: Design a helmet practice project** four-part video series. Review these instructions to become familiar with the suggested steps as you prepare to guide students though completing this project using both Tinkercad and Fusion 360. Or, if you're creating the helmet design on your own, follow the step-by-step instructions to make your design.

Each section of this guide includes learning objectives, key terminology covered in the videos, and instructions with screenshots for your reference.

View the entire four-part series on our **Level Up from Tinkercad to Fusion 360** playlist: *https://bit.ly/3nMmTbQ*

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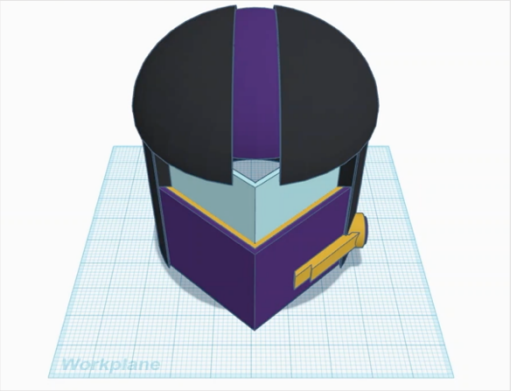
Step-by-step: Starting a design in Tinkercad (Part 1 of 4)

This portion of the step-by-step guide is designed to accompany the **Level Up Helmet Project (Part 1 of 4): Starting a design in Tinkercad** video.

Link to Video 1:
https://bit.ly/3FQbHRC



Basic helmet design in Tinkercad

Learning objectives

After completing the steps outlined in this guide using Tinkercad, you should be able to:

- Create 3D objects.
- Connect objects to each other.
- Measure objects.
- Review connections between objects.

Key terminology

- **Align**: A tool in Tinkercad that lets you place or arrange objects in a straight line. To use the Align tool, select at least two objects by Shift left-clicking on them or by dragging a box around them. Once selected, click on the Align icon at the top. Simply move your mouse over a node (the black dots) to preview the move.
- **Computer Aided Design (CAD)**: A type of computer software that allows you to create multi-dimensional objects
- **CAD model**: A design created in CAD software. Examples of CAD software include Autodesk's Tinkercad and Fusion 360, which allow you to create 3D representations of your ideas
- **Design parameters**: Values that are assigned to something you make, like size, weight, and materials
- **Duplicate**: A tool that lets you make an exact copy of an object. To duplicate an object, use Ctrl + D and then drag it out or use the arrow keys
- **Group**: A tool in Tinkercad that lets you combine two or more shapes into a part. Do this by selecting them and then choosing the Group icon at the top
- **Hide**: A tool that lets you to save the objects and clear the workspace
- **Hole**: A tool used to subtract from a solid shape
- **Ruler**: Use this tool to display or change the dimensions or distance of a shape or object
- **Viewcube**: A cube-shaped tool that lets you rotate your perspective for view specific views, including front, side, right, left, top, and bottom
- **Workplane**: The large, blue grid where you create your designs in Tinkercad. You can drag out new workplanes onto the surfaces of your shapes for easier stacking and more precise measuring

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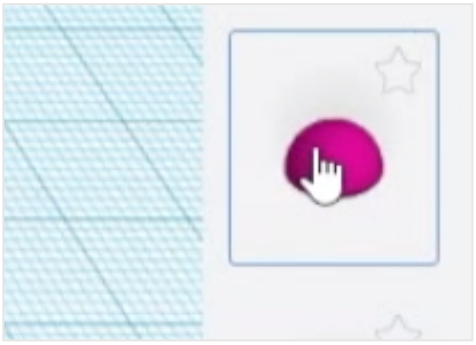
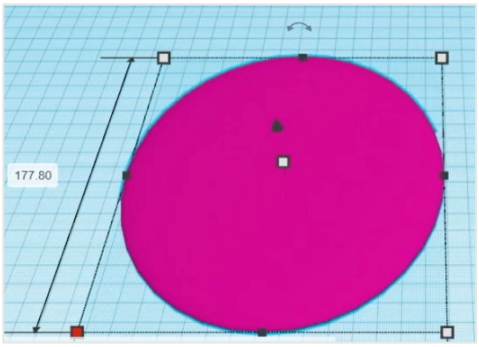
Step-by-step instructions

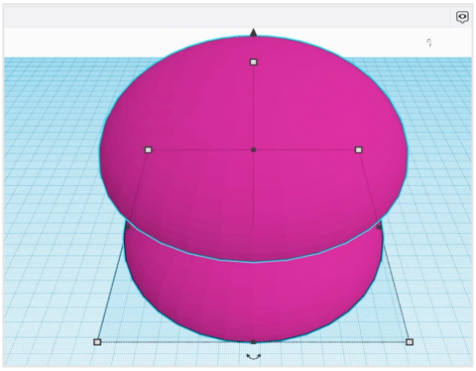
1. Open a new Tinkercad file. Start your design by creating the base of the cap, which is a simple dome-like shape. Select a **Half Sphere** from the Basic Shapes Toolbar.



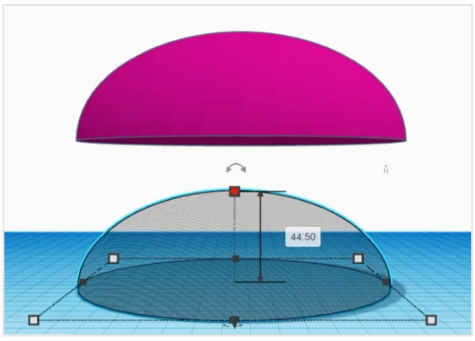
2. Modify the Half Sphere into an oval shape on the workplace using these dimensions. To switch the type of unit you want to use, click **Edit Grid** and then either **Millimeters or Inches** from the dropdown menu.
   - Width: 152.4 mm or 6 inches
   - Length: 177.8 mm or 7 inches
   - Height: 50.8mm or 2 inches



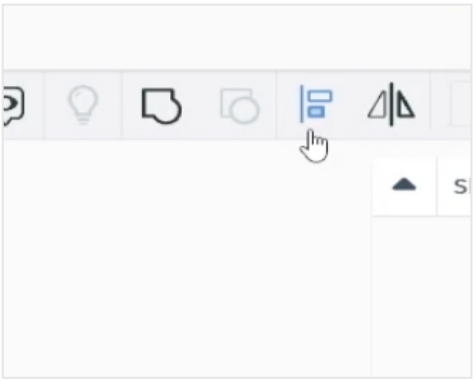
3. Duplicate the shape by selecting it and then selecting the **Duplicate** button, or by holding down the **CTRL + D** keys.



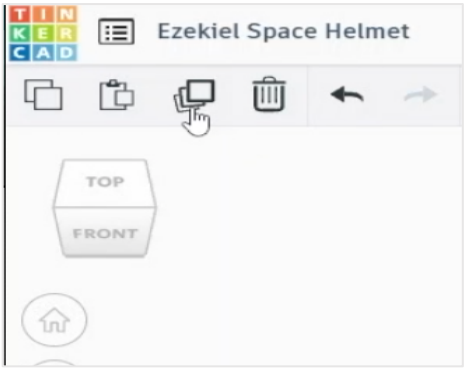
4. Make sure you select the shape, and then change the duplicate shape into a hole by selecting the **Hole** button in the Shape Toolbar. Then change the Hole Shape dimensions:
    - Width: 146 mm or 5.75 inches
    - Length: 171.5 mm or 6.75 inches
    - Height: 44.5 mm or 1.75 inches



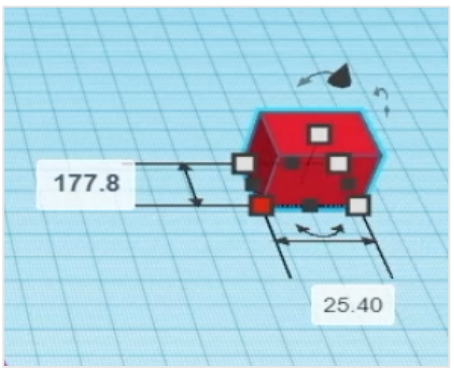
5. Next, select both shapes and use the **Align** tool to overlap both shapes, with the hole shape inside the solid shape.



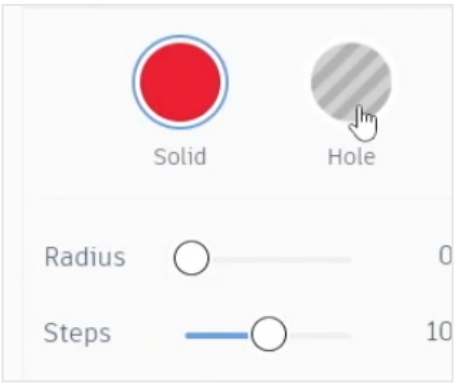
6. Select both shapes and use **Group** tool, or hold down the **CTRL + Shift + G** keys, to create a hollowed single shape. Then select the new shape and use the **Duplicate** tool to create two.



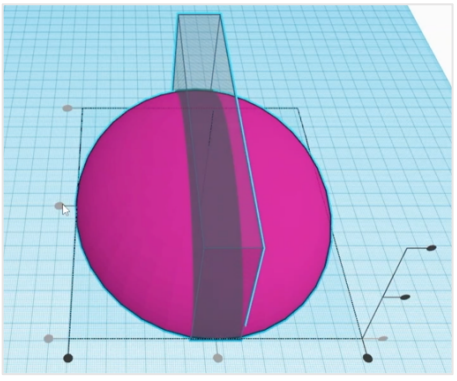
7. Now, drag a Box shape onto the workplane and change its dimensions:
    - Width: 25.4 mm or 1 inch
    - Length: 177.8 mm or 7 inches
    - Height: 76.2 mm or 3 inches



8. Turn the box into a hole by selecting the **Hole** button in the Shape toolbar.

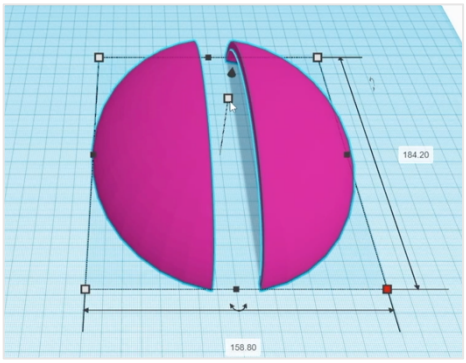


9. Select both shapes and use the **Align** tool to overlap. The hole should run down the center of the oval shape.



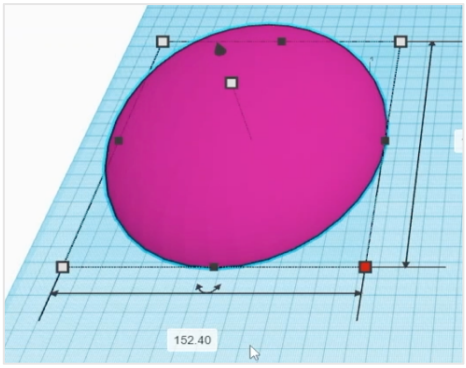
10. Next, select the shape with the hole running through it and modify its dimensions:

- Width: 158.8 mm or 6.25 inches
- Length: 184.2 mm or 7.25 inches
- Height: 44.5 mm or 1.75 inches



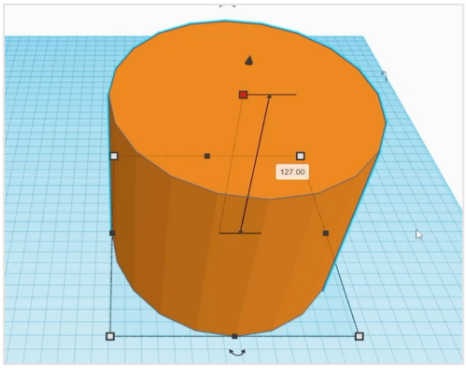
11. Now change the other duplicated shape's dimensions, and then select **Hide** to clear the workspace from view.

- Width: 152.4 mm or 6 inches
- Length: 177.8 mm or 7 inches
- Height: 44.5 mm or 1.75 inches



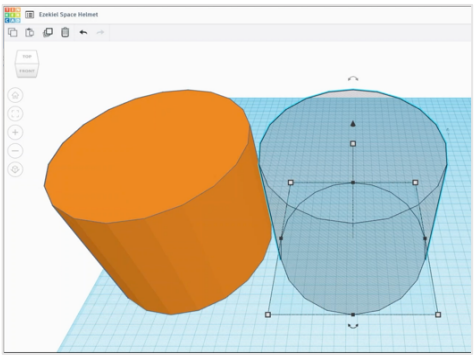
12. To get started on the next component of the model, drag a Cylinder shape to the workplane, and modify its dimensions:

- Width: 152.4 mm or 6 inches
- Length: 177.8 mm or 7 inches
- Height: 127.0 mm or 5 inches

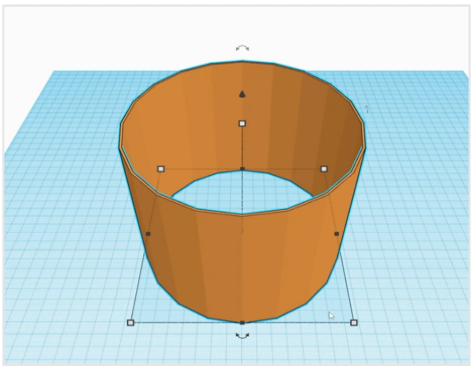


13. Duplicate the shape to create two shapes, and turn one of the shapes into a hole. Then modify the hole shape's dimensions:

- Width: 149.9 mm or 5.9 inches
- Length: 175.3 mm or 6.9 inches
- Height: 127 mm or 5 inches

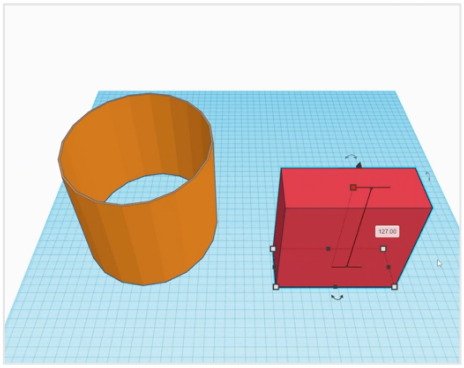


14. Select both shapes, use the **Align** tool to overlap then, and then select **Group**. The cylinder is now hollow.

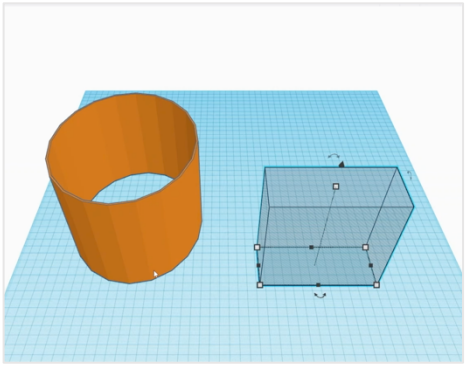


15. Next, drag a box to the workplane, and modify its dimensions:

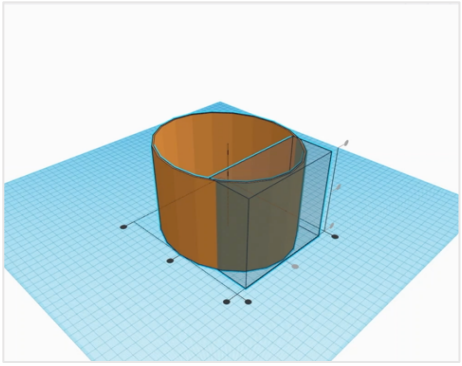
- Width: 127 mm or 5 inches
- Length: 50.8 mm or 2 inches
- Height: 127 mm or 5 inches



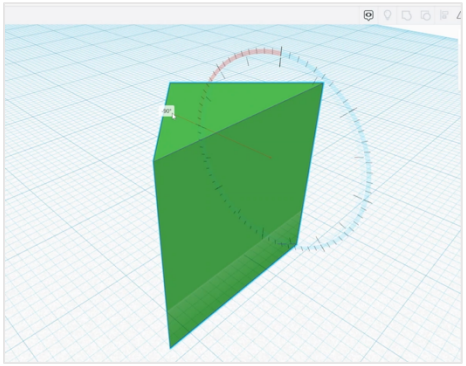
16. Convert the shape into a hole with the **Hole** tool.



17. Now overlap the rectangular box-shaped hole into the hollowed cylinder in a way that creates an opening widthwise across the cylinder. To do this, select both shapes and use the **Group** tool to merge them. Then select **Hide** to save the objects and clear the workspace.

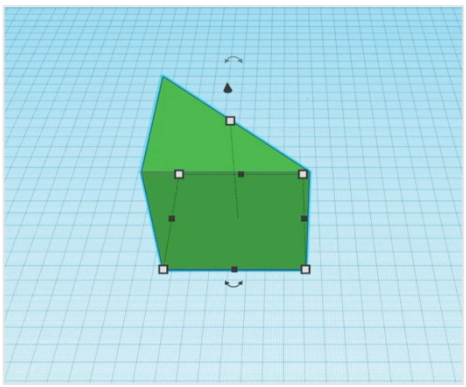


18. Now that the base is complete, start the visor by dragging a Roof shape to workplane, and then rotate it 90 degrees so it's facing upward. Rotate it once more 45 degrees counterclockwise.

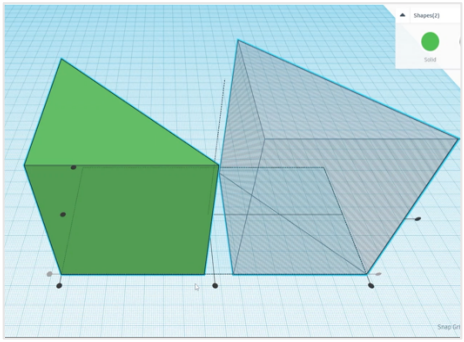


19. Modify the dimensions of the roof shape:

- Width: 76.2 mm or 3 inches
- Length: 76.2 mm or 3 inches
- Height: 76.2 mm or 3 inches

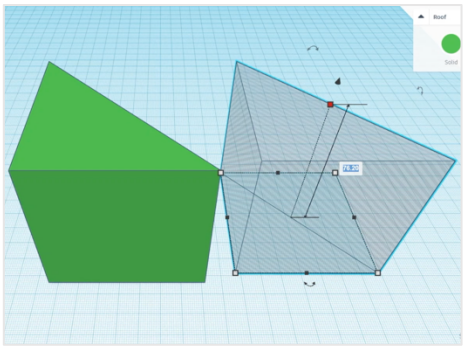


20. Duplicate the shape to create two, and then select one of the roof shapes and use the **Hole** tool to turn it into a hole.

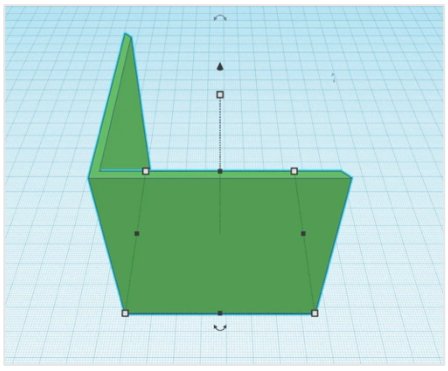


21. Change the dimensions:

- Width: 71.1 mm or 2.8 inches
- Length: 71.1 mm or 2.8 inches
- Height: 88.9 mm or 3.5 inches

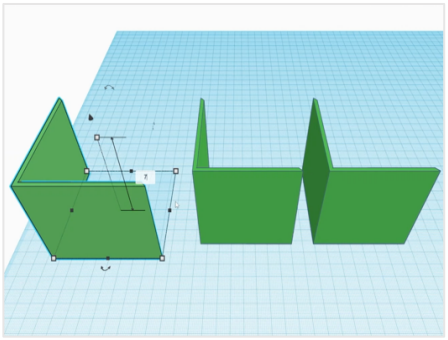


22. Select both shapes. Use the **Align** tool to overlap both shapes and the **Group** tool to group them together.

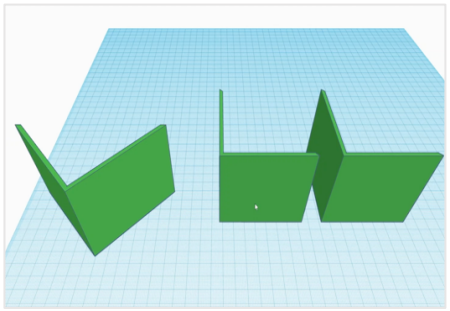


23. Duplicate the shape to create three identical shapes. Then select the first duplicate and change its dimensions.

- Width: 88.9 mm or 3.5 inches
- Length: 88.9 mm or 3.5 inches
- Height: 76.2 mm or 3 inches

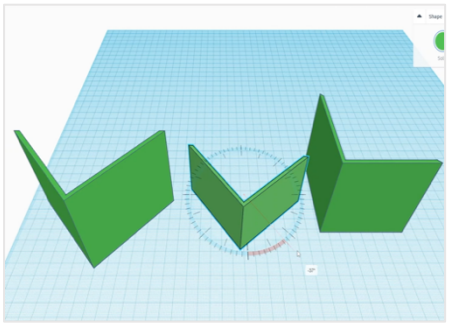


24. Rotate the shape 45 degrees counterclockwise.



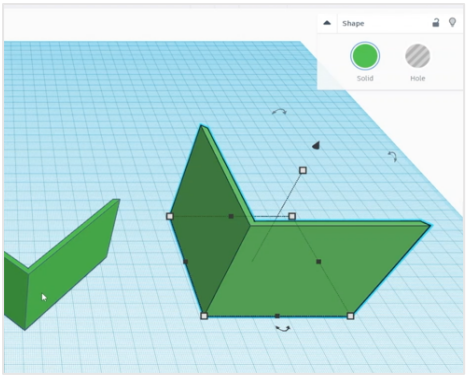
25. The next step is to select the second duplicate and change its dimensions. Then rotate this shape counterclockwise like the other shapes.

- Width: 76.2 mm or 3 inches
- Length: 76.2 mm or 3 inches
- Height: 50.8 mm or 2 inches



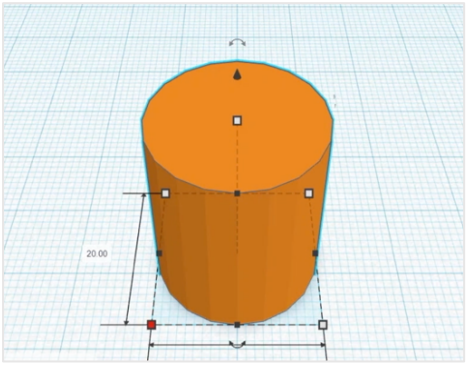
26. Repeat this process with the third duplicate on the workplane, using these dimensions. Then, hide the objects and clear the workspace once more.

- Width: 82.6 mm or 3.25 inches
- Length: 82.6 mm or 3.25 inches
- Height: 12.7 mm or 0.50 inch



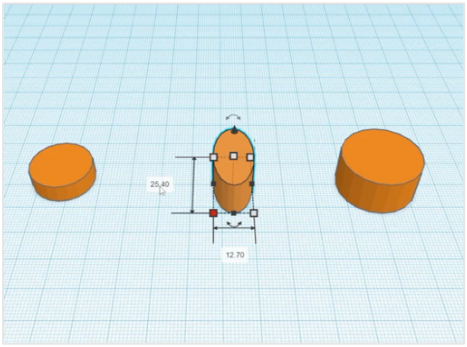
27. For the final component, you'll create a screw to hold the visor on the helmet. To start, drag a Cylinder to the workplane and change its dimensions:

- Width: 25.4 mm or 1 inch
- Length: 25.4 mm or 1 inch
- Height: 12.7 mm or 0.50 inch



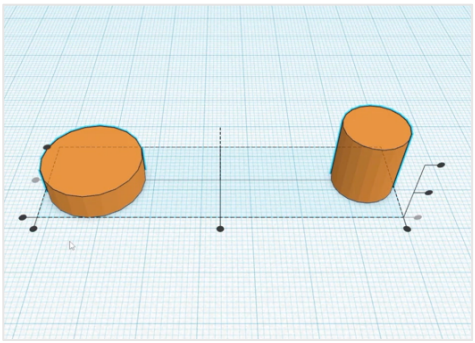
28. Duplicate the shape to create three cylinders, and change one of the shapes to these dimensions:

- Width: 20.32 mm or 0.80 inch
- Length: 20.32 mm or 0.80 inch
- Height: 6.35 mm or 0.25 inch

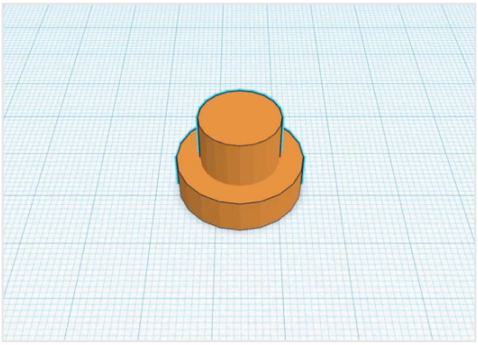


29. Next, modify another one of the cylinders into the following measurements:

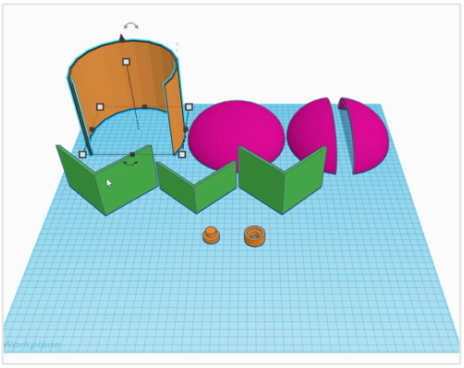
- Width: 12.70 mm or 0.50 inch
- Length: 12.70 mm or 0.50 inch
- Height: 15.2 mm or 0.60 inch



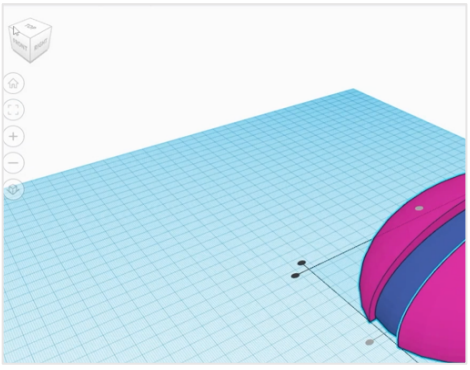
30. Now select both previously modified shapes and align and group them. See how it looks like a bolt screw?



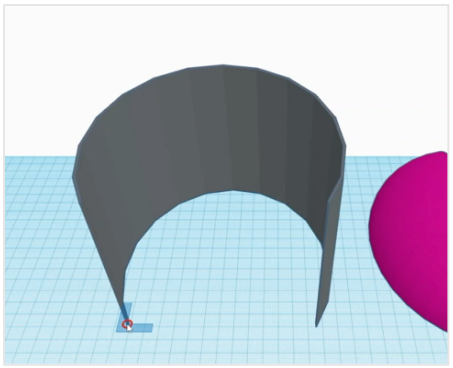
31. Now that you have all the parts designed, it's time to assemble. To start, select **Hide** to reveal the previous components for final assembly.



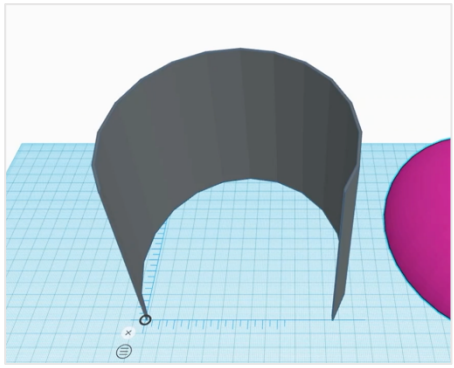
32. Use the **ViewCube** to make sure your components are touching each other so that no components end up floating in space.



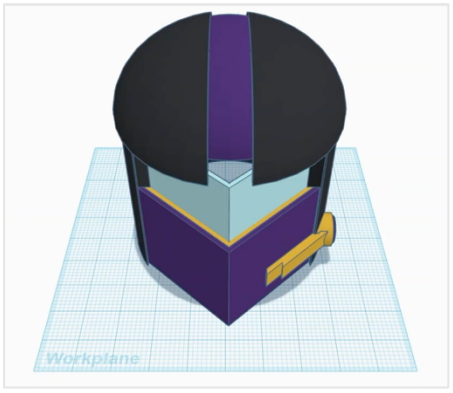
33. Use the **Ruler** tool to ensure your components are exactly where you want them to be, and then measure. Place the Ruler at the edge of the helmet base.



34. To change the type of measurement you want, like the distance from one object endpoint to another, click the circle at the corner of the ruler.



35. Feel free to make tweaks to your design, like color edits. Now you have a basic model!



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Step-by-step: Sending a design from Tinkercad to Fusion 360 (Part 2 of 4)

This portion of the step-by-step guide is designed to accompany the **Level Up Helmet Project (Part 2 of 4): Sending a design from Tinkercad to Fusion 360** video.

Link to Video 2:
https://bit.ly/33CzgAo

Basic helmet design in Fusion 360

Learning objectives

After completing the steps outlined in this guide, you should be able to use Tinkercad and Fusion 360 to:

- Send a CAD model from Tinkercad to Fusion 360.
- Open an exported file in Fusion 360.
- Navigate the basic features of the Fusion 360 interface.

Key terminology

- **Assembly**: Composed of one or more components
- **Body:** A continuous 3D object; the Bodies dropdown menu is listed under each component
- **Browser**: The area where any objects in a CAD model or assembly are listed, which lets you keep track and control objects in an assembly
- **Component**: Composed of one or more bodies
- **Data Panel**: The area that shows your data or hides your data if it's overwhelming the workspace
- **My Profile**: The area where you can view your account and sign out
- **Send to Fusion 360**: After selecting all the parts of a design, this button lets you prepare your model to open in Autodesk's Fusion 360
- **Timeline**: Allows you to keep track of each step you've completed in your design process by showing all operations performed in order. If sending a design to Fusion 360, the "Capture Design History" option must be chosen
- **Toolbar**: Organizes a wide variety of tools available in Fusion 360
- **Workspace Picker**: The area that allows you to choose the appropriate workspace in Fusion 360

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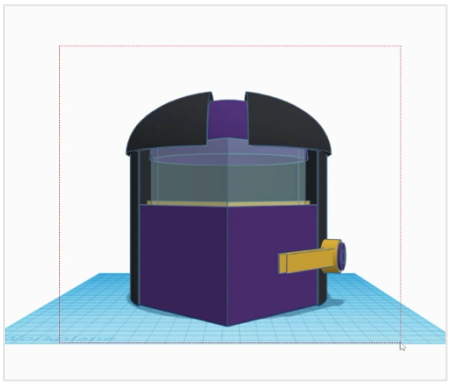
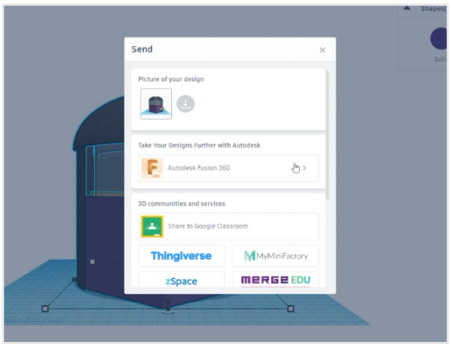
Step-by-step instructions

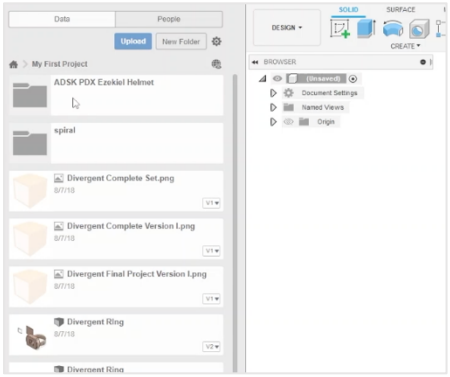
1. Prepare to export your Tinkercad file by making sure all parts are selected.



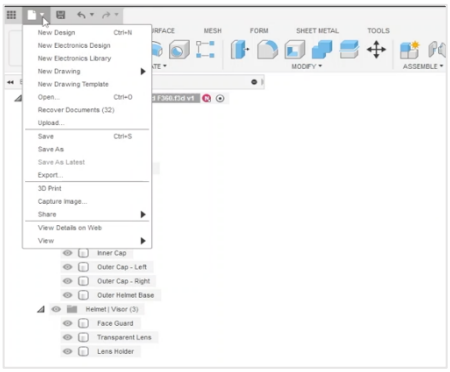
2. Select the Send to Fusion 360 button at the top right of the screen. Your file is then ready to open in Fusion 360.



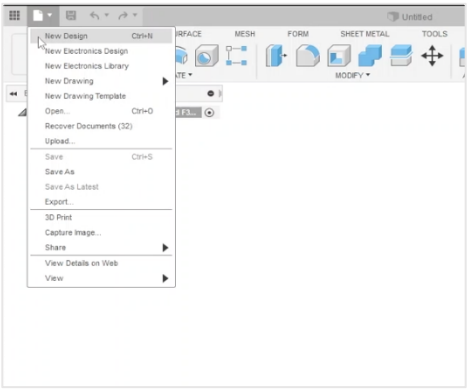
3. Take some time to navigate the new design environment in Fusion 360. Select the **Data Panel** to review your data. You can also use this tool to hide your data if it's overwhelming your workspace.



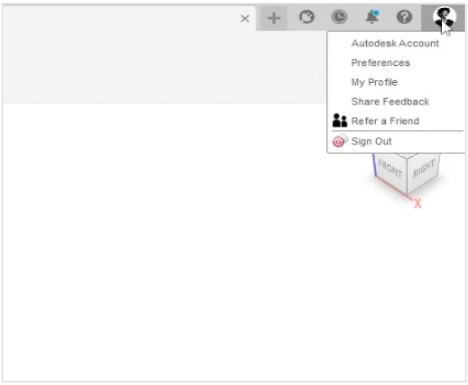
4. Next, navigate to the **Application Bar**. Similar to word processing software, here you'll find the **File** menu as well as **Save** and **Undo/Redo** function.



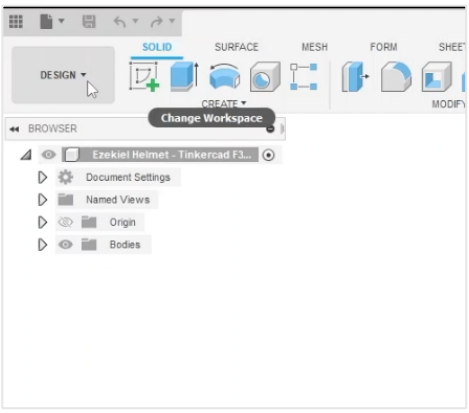
5. The **New Design** button lets you create a new design.



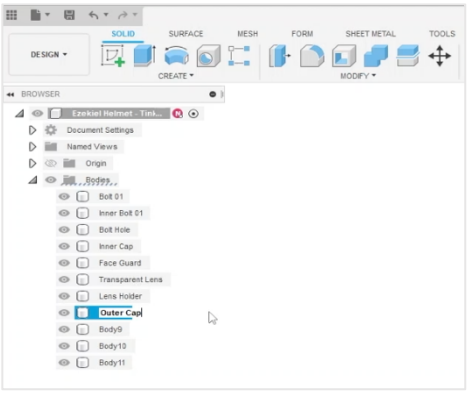
6. You can view your account and sign out in the **My Profile** area, similar to Tinkercad.



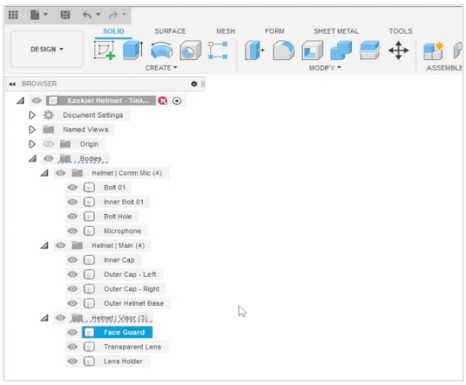
7. Now, take a few moments to explore the **Toolbar**. In contract with Tinkercad, where all your tools fit on a single toolbar above your workspace, Fusion 360 has an enormous number of tools. Just as it's easier to find a tool like a spatula in a kitchen workspace than a garage workspace, Fusion 360 offers various **Workspaces** to make it easier to organize your work. You can switch your workspace using the **Workspace Picker**.



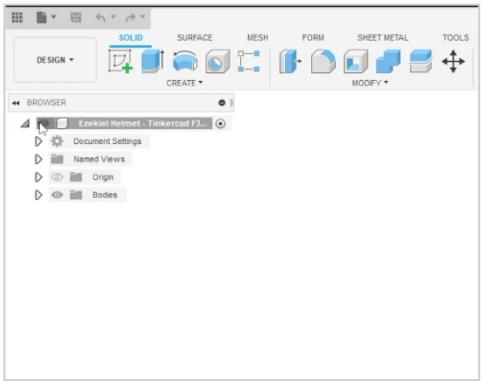
8. Take some time to explore the **Browser**. This is where any objects in your CAD model or assembly will be listed. It lets you keep track of as well as control objects in your assembly.



9. Anything that moves independently in your design should be its own **Component**. For example, in the helmet design, your assembly is your helmet, which is composed of a top cap, face guard, mount, etc. In turn, each component is made up of one or more **bodies** or **continuous 3D objects** you created in Tinkercad.

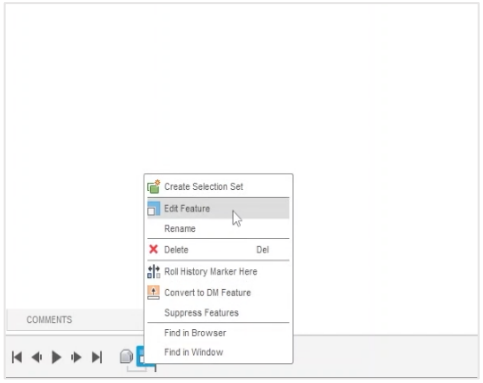


10. If you look you at the Browser, you'll see why it's important to know the difference. When you imported your helmet CAD model, several components were already created. The top-level file is the assembly containing all the components. You'll also notice the **Bodies** drop-down menu under each component. This area lists any bodies associated with a particular component or assembly.



11. Now turn your attention to the **Timeline**, which allows you to keep track of each step you've done in the design process. Any operation performed on your design will be listed here in order.

Now that you can navigate your way around Fusion 360, next you can start bringing your helmet design to life!

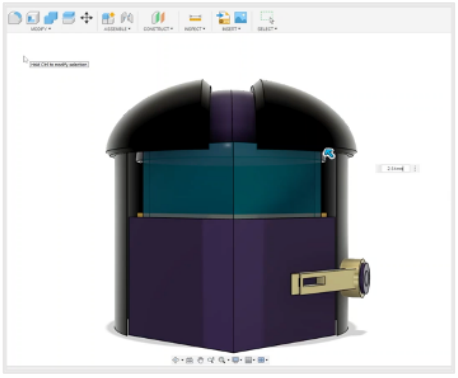


AUTODESK

Step-by-step: Modifying a Design in Fusion 360 (Part 3 of 4)

This portion of the step-by-step guide is designed to accompany the **Level Up Helmet Project (Part 3 of 4): Modifying a design from Tinkercad to Fusion 360** video.

Link to Video 3:
https://bit.ly/3qR9AsR



Modify a helmet design in Fusion 360

Learning objectives

After completing the steps outlined in this guide, you should be able to use Fusion 360 to:

- Enhance or refine a basic design created in Tinkercad.
- Round Tinkercad's hard edges by adding fillets.
- Emphasize the smoothness of shapes in Fusion 360.
- Bevel edges by adding chamfers.

Key terminology

- **Canvas**: The area where you design your project, similar to the workplace in Tinkercad
- **Chamfer**: A command that removes material from a sharp object to create a blunt edge or bevel
- **Extrude**: A command that lets you transform a sketch or a face by adding depth, including creating a new solid body, joining to an existing body, removing material from an existing body, or creating a new component
- **Fillet**: A command that lets you round the edges of a solid body
- **Home**: The button that lets you return to your default view
- **Modify**: A menu that offers a variety of options for changing your design, including Press Pull, Fillet, Chamfer, Shell, etc.
- **Press Pull**: A command that lets you change the size or geometry of faces, bodies, and edges
- **Sketch**: A 2D blueprint that outlines the overall shape and finer details of the features you want to create in your design

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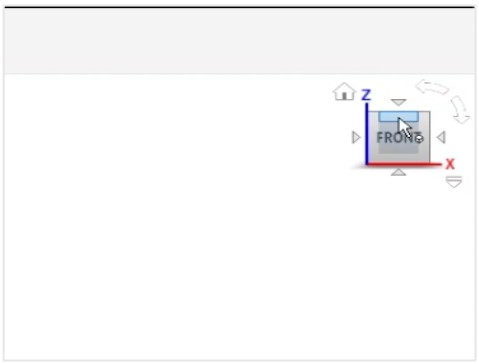
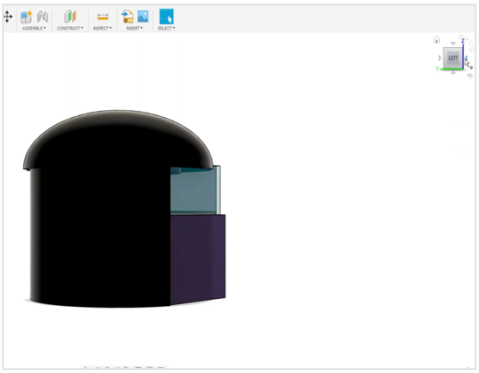
Step-by-step instructions

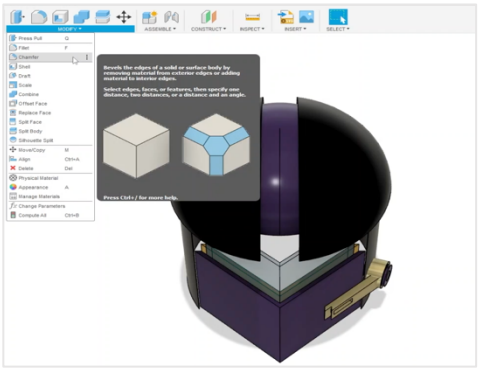
1. Click the front face of the **ViewCube** in the upper-left corner of the **Canvas**, which is what we call the **Workplane** in Tinkercad. The front of the helmet should face you. To orbit the CAD model in your canvas, left-click and drag the ViewCube.



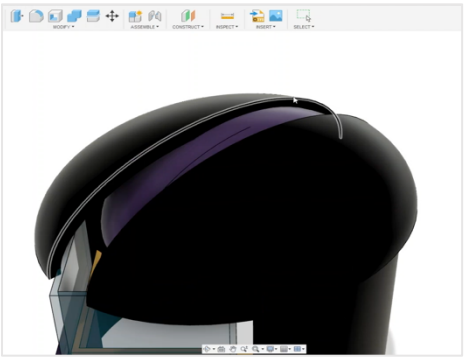
2. For more precise controls, click the corners of the ViewCube to rotate the camera and see some isometric views. To return to your default view, click **Home**.



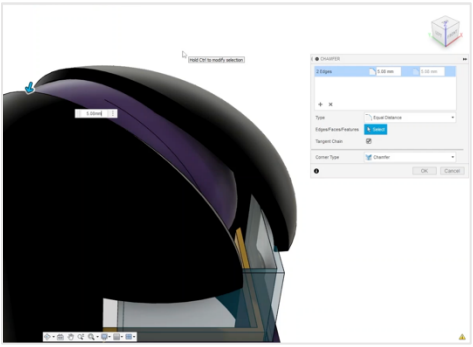
3. At the top of your helmet design you'll notice there are two halves of an oval. Since you imported the components of your CAD model from Tinkercad, you can use the **Modify** drop-down menu to change your design. To make your first modification, select **Chamfer** from the Modify drop-down menu.



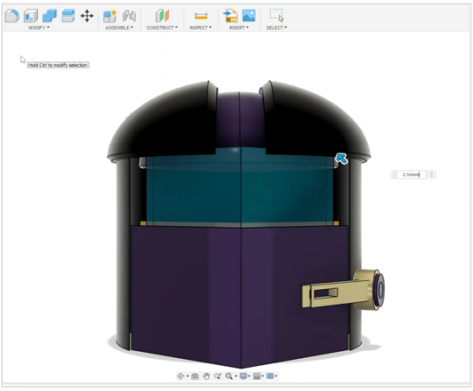
4. The **Chamfer** command removes materials from the object to create a blunt edge or bevel on the edges of a sharp object. To do this, select the outer inside edge of one of the half ovals on the top of the helmet, and then select **Chamfer**.



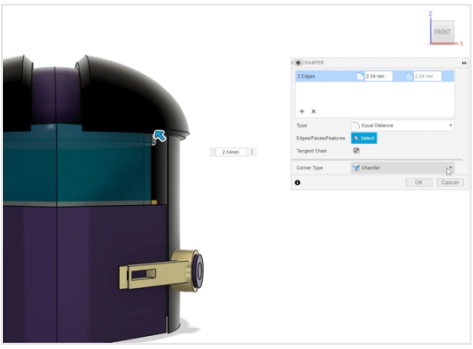
5. Now you'll see the Chamfer dialog box. Here, select **Equal Distance** as the Chamfer Type. Modify the **Corner Type** setting to **Chamfer** to have the newly beveled edges meet at the corner. Keep the other options at the default settings, except corner types, which you'll modify to Chamfer to have your newly beveled edges meet at the corner. Modify the distance of the selected edge by 5.08 millimeters, or .2 inch. Then select **OK**. The first edge is now chamfered.



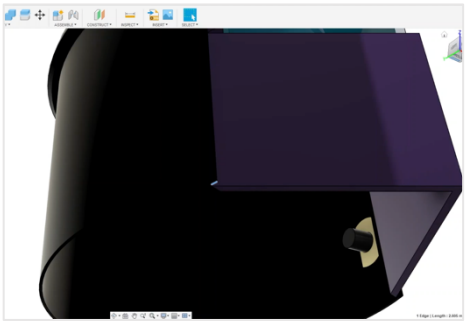
6. Select both of the upper outer edges of the shape that curves around the top of the helmet. Once you select one, hold the **Shift** key and the similar bottom edge on the opposite side to select multiple objects at once. Next, re-select the **Chamfer** command from the **Modify** menu. In the **Chamfer** dialogue, modify the selected edges by 2.54 millimeters, or a tenth of an inch. Repeat this process elsewhere on your helmet, as needed.



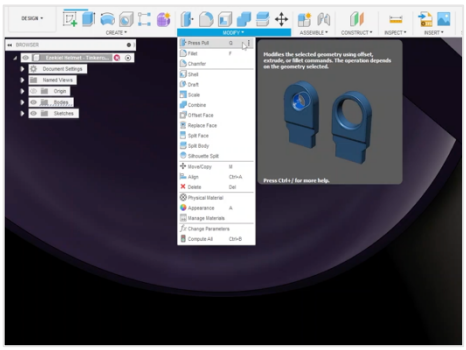
7. Reorient the helmet by clicking the bottom front face of the ViewCube in the upper-left corner. The front of the helmet should face you for this next step.



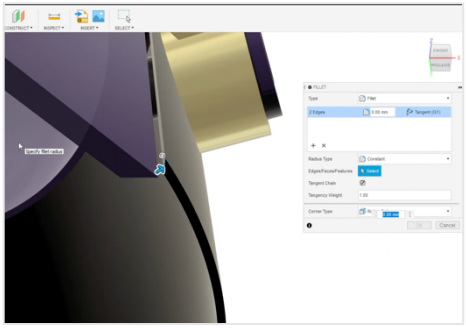
8. Look for the bottom edge along the thickness of the visor forming the cylindrical base of the model on both sides. After selecting one, hold the **Shift** key and find the similar bottom edge on the opposite side. This lets you select multiple objects at once.



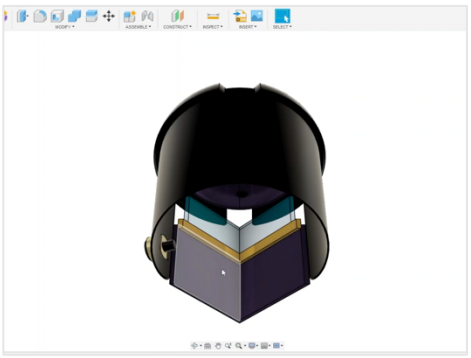
9. Now you can modify the edges by using the **Press Pull** command from the **Modify** menu, which lets you change the size or geometry of faces, bodies, and edges.



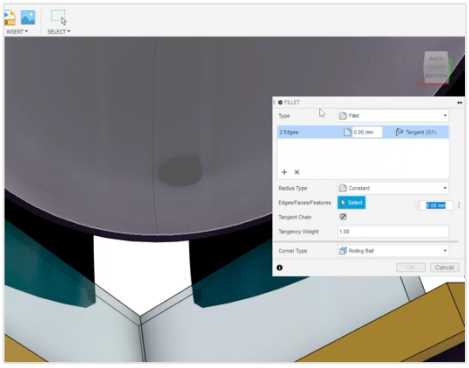
10. Because you selected the edges, the **Press Pull** command automatically routes you to the **Fillet** command. This allows you to round the edges of a solid body. In the dialogue box, insert a fillet radius of 11.43 millimeters, or .45 inch on the selected edges.



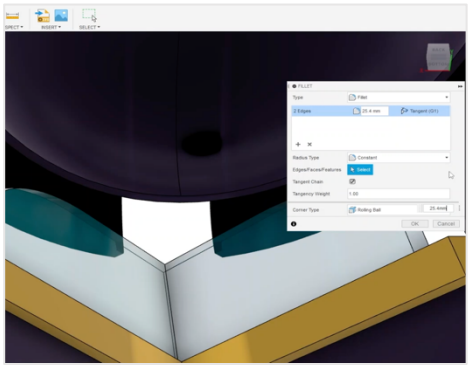
11. Reposition the ViewCube to the bottom face so that you see a view of the interior of your model, including the transparent lens component.



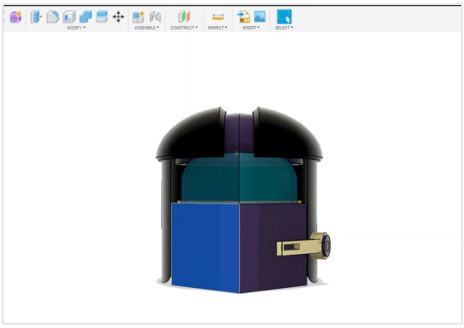
12. Next, use the **Fillet** command to add additional rounded edges. In the dialog, select the edge along the width of the lens. Then hold the **Shift** key while selecting the edge on the opposite side.



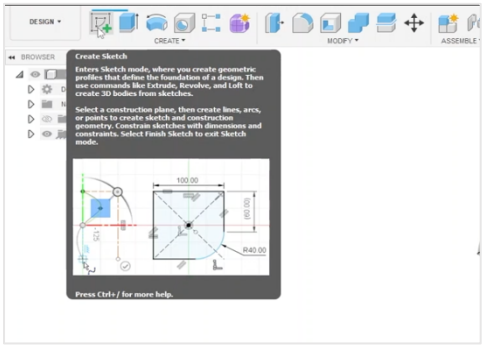
13. Select the **Modify** menu and choose the **Fillet** command in the dialog. Modify the Radius type to **Constant** to choose a consistent edge. Then, round this up by modifying selected edges by a radius length of 25.4 millimeters, or 1 inch. Close the dialog by clicking **OK**.



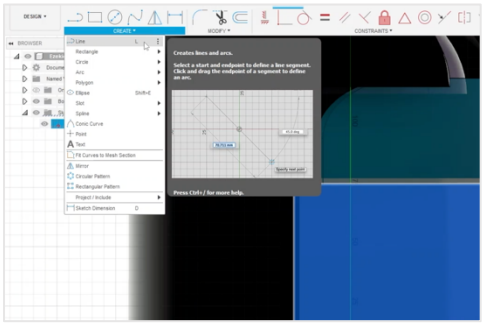
14. Now that you've used the **Chamfer** and **Fillet** commands, explore some other features of Fusion 360 to refine your design. To add new features to your face guard, first activate the Face Guard Component by selecting it in the canvas or selecting the component in the Browser.



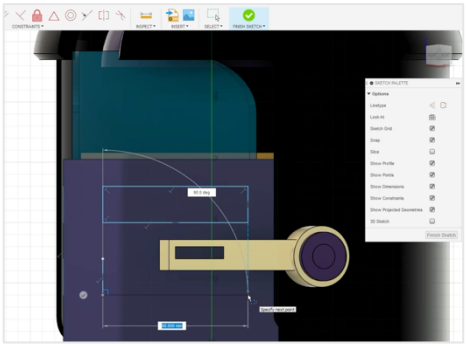
15. Now it's time to create a **Sketch,** which is a 2D blueprint that outlines the overall shape and finer details of the features you want to include. Every sketch requires a 2D face or plane to draw on. Click the **Create** panel and select the **Create Sketch** command.



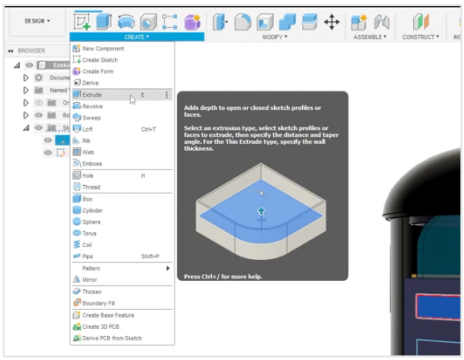
16. Select the **Line** command under the **Create** menu and sketch a shape of your choosing by connecting various lines. Use the **Line** command to create various rectangular shapes. When you're done, click the **Finish Sketch** button.



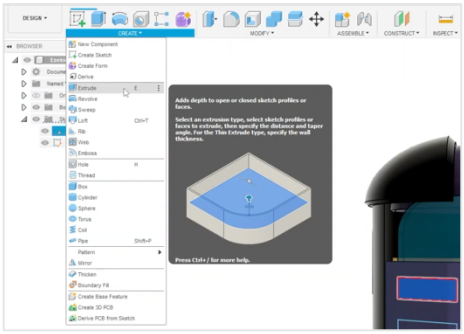
17. Repeat the process on the opposite side of the visor. To locate your recent sketches, look in your **Browser**.



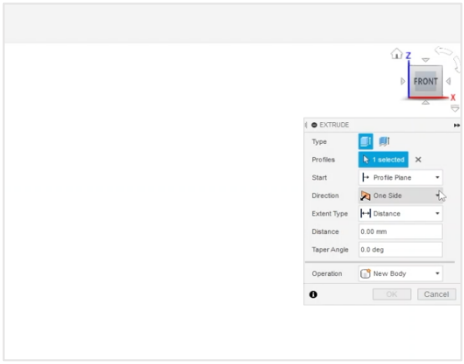
18. Now that you've completed your sketches, it's time to extrude them so you can transform your sketches or faces by adding depth. The **Extrude** command can be used to create a new solid body, join to an existing body, remove material from an existing body, or create a new component. Start by selecting one of the sketches you created.



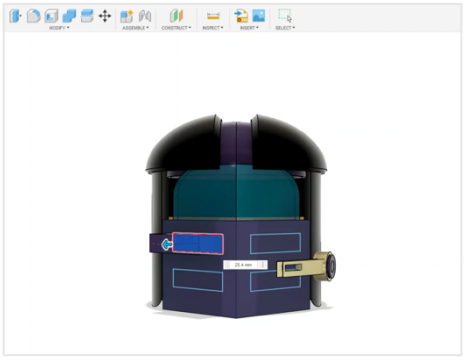
19. Click **Create Menu** and select the **Extrude** command. When the dialog appears, select **Entire Area** as Extrude Type.



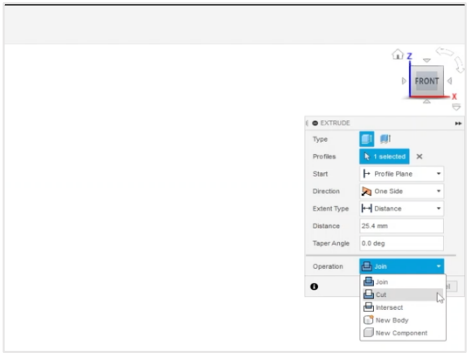
20. Next, select **Start** as Profile Plane, which means the extrusion will start from the face or the profile plane you've selected.



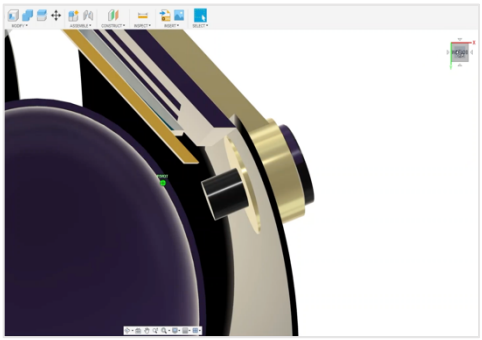
21. Now select the **Direction** as **One Side**. Then modify the **Distance** or depth of the extrusion as 25.4 millimeters, or 1 inch.



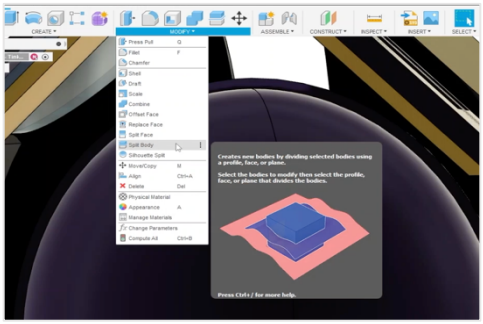
22. There are a few Operations you can perform in the **Extrude** dialog. **Join** lets you combine the new shape you're creating with an existing body, like your face guard. **Cut** lets you cut an area out of an existing body based on the new body. Let's choose **Cut** here. Select OK to exit out of the Extrude dialog. The process repeates for the remaining sketches on the face guard. Let's move on to the helmet mount.



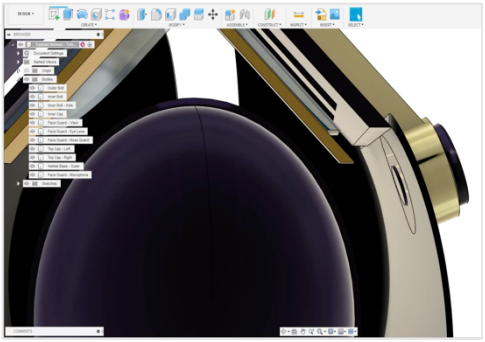
23. Use the ViewCube to orient the helmet so you can see the bottom face of the helmet mount. Then activate the helmet mount by selecting it in the canvas or selecting the component in the Browser. It should overlap the cylindrical base.



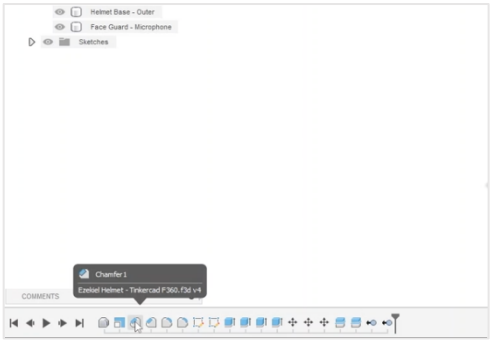
24. Now click the **Modify** menu and select the **Split Body** command. In the dialog, select the outer face of the helmet's cylindrical base as the **Splitting Tool**. Then deselect **Extend Splitting Tool** and choose **OK.**



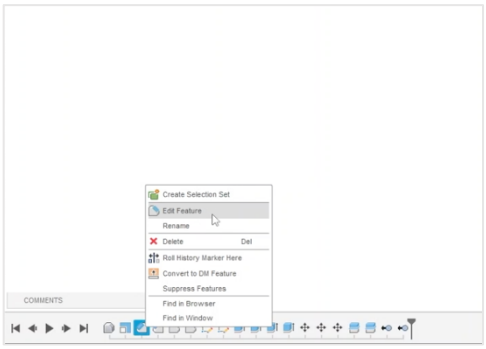
25. Next, delete the remaining parts you cut using the **Splitting Tool**. Now you have a complete Fusion 360 model!



26. If there's something you'd like to change in your design, you can use Fusion 360's "CAD time machine," otherwise known as the **Timeline**, in the lower-left corner.



27. The timeline lets you scroll past your assembly timeline. You can see every command, including the **Chamfer** command, you've completed so far. You can make edits as needed by right-clicking the **Chamfer** command, which opens the dialog. Adjust the Chamfer distance from 5.08 millimeters to 6.35 millimeters.



28. Next, When your edits are complete, you can click **Play** on the Timeline to see how each command has elevated your original design from Tinkercad.

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Step-by-step: Rendering a Design in Fusion 360 (Part 4 of 4)

Exercise duration: 30 minutes

This portion of the step-by-step guide is designed to accompany the **Level Up Helmet Project (Part 4 of 4): Rendering a design in Fusion 360** video.

Link to Video 4:
https://bit.ly/3tKqXNK
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Render a helmet design in Fusion 360

Learning objectives

After completing the steps outlined in this guide, you should be able to use Fusion 360 to:

- Modify appearances for a design.
- Add environments to a render.
- Create a cloud render.

Key terminology

- **Appearance:** A tool that lets you change the material used on a body or face, including transparent plastic, metal, wood, and more.
- **Decal**: A tool that lets you apply a simple image on a component or body, such as a logo or text
- **Cloud rendering:** Uses the internet to produce high-quality images of a canvas
- **Render Workspace**: The workspace that allows you to create high-quality, photo-realistic images that can be used to showcase a design
- **Render**: A tool that lets you create photo-realistic images of a design
- **Rendering Gallery:** An area that displays thumbnails of your in-progress and complete cloud renderings
- **Scene Settings**: A tool that lets you change the lighting of a design or the environment where you set your design
- **Texture**: A tool that lets you add photo-realistic details to the appearance of design

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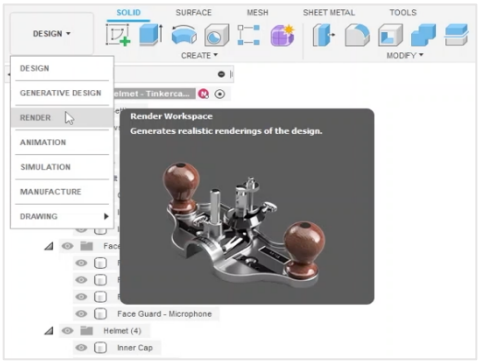
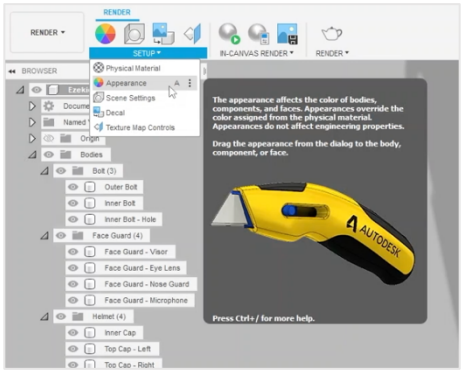
Step-by-step instructions

1. In the upper-left corner of the window, use the **Workspace Picker** to transition to the **Render Workspace**, where you can create high-quality images to communicate your ideas.

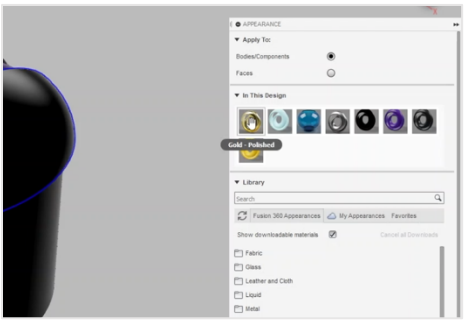


2. In the **Set Up** panel, choose **Appearance.**

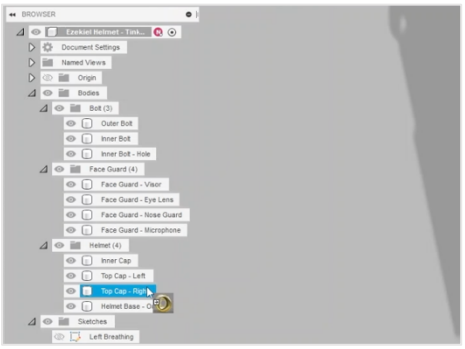


3. There are two main methods to add materials to bodies and faces from the **Appearance** dialog. The first step of the first method is to select the surface/face of a body you want to modify.

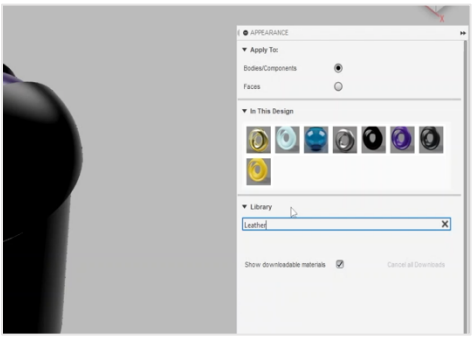
4. Next, select a material from the **Library** and drag it from the **Appearance** dialog. (Note: This method only applies the selected material and finish to the pre-selected face.)



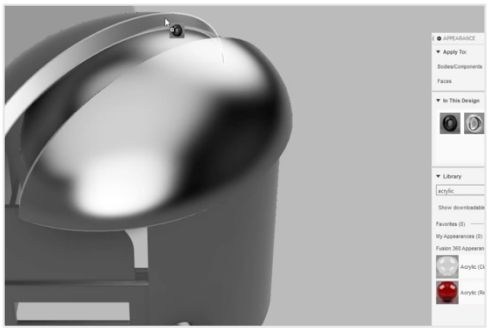
5. The second method to add materials to components and bodies is to highlight the body you wish to modify in the **Browser**. Then select and drag a material from the **In This Design** area in the **Appearance** dialog to the highlighted compnent or body in the **Browser**. This method applies the selected material and finish to the entire body, and not just the face.



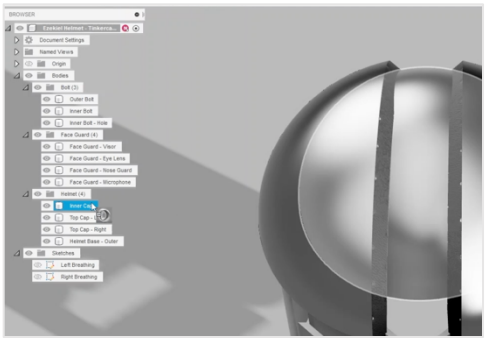
6. Choose the following materials for the **In This Design** folder within the **Appearance** dialog:
    - Leather
    - Aluminum – Polished (Metal)
    - ABS (white)
    - Plastic – Glossy (black)
    - Walnut
    - Acrylic (clear)



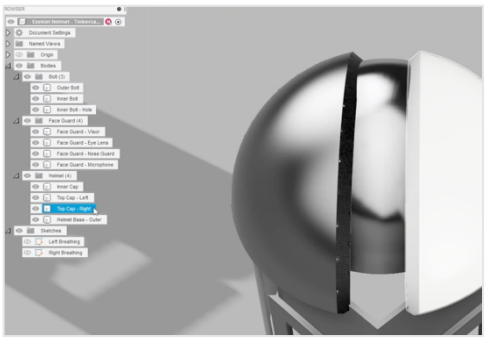
7. Use the first method on the outer-curved surface of the helmet's top cap to apply the **Leather** material.



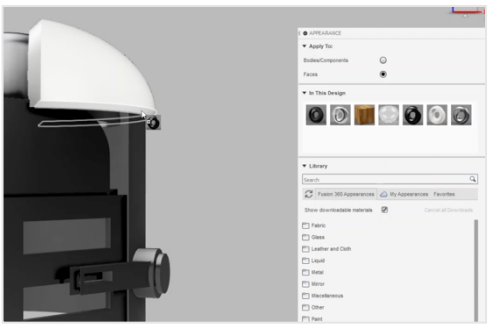
8. Use the second method on both halves of the split top to apply the **Aluminum** material.



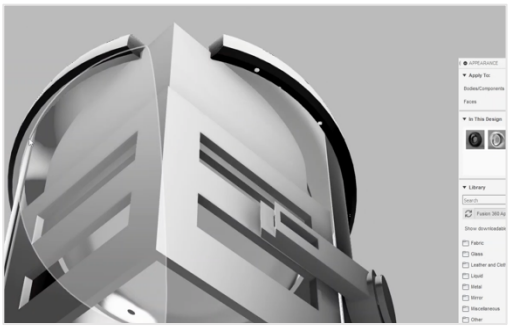
9. Use the second method once more on both halves of the split top cap to apply another material, this time from the plastic family: ABS (white).



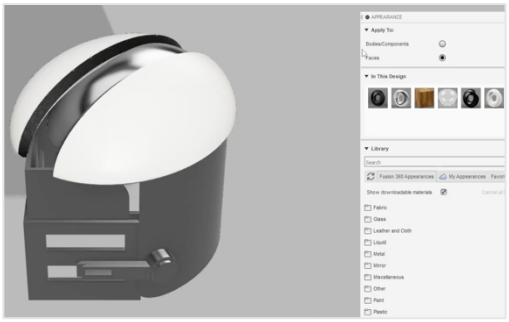
10. Now, using the first method on both tops of the outer curve of the helmet's top cap split, apply the Plastic – Glossy (black) material.



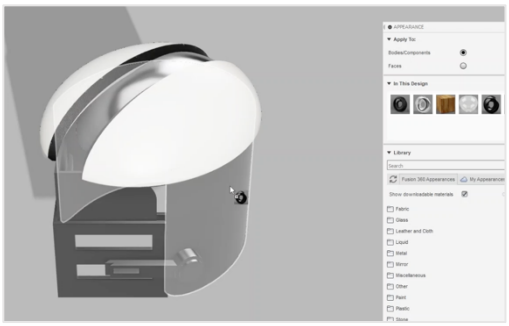
11. Then on both lower-bottom curves of the top cap split, apply the **Leather** material.



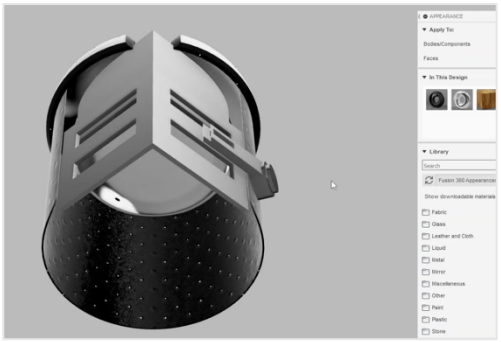
12. On the cylindrical base, apply **ABS (white)**.



13. On the outline face of the cylindrical base, apply the **Plastic – Glossy (black)** material.

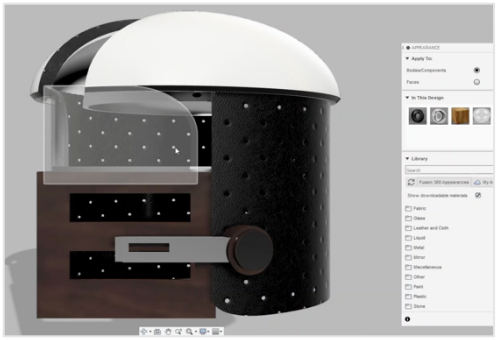


14. Now on the inside face of the cylindrical base, apply **Leather**. And voila, most of your helmet is now complete!

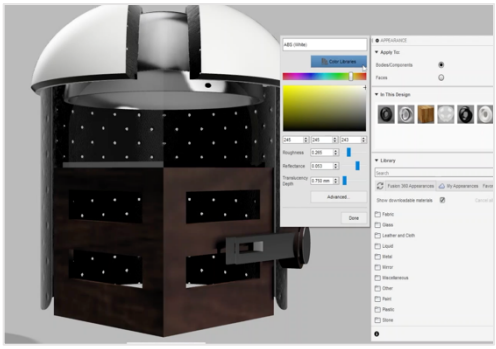


15. Add some finishing touches now. Add the **Walnut** material to the face guard and the bolts of the helmet mount.

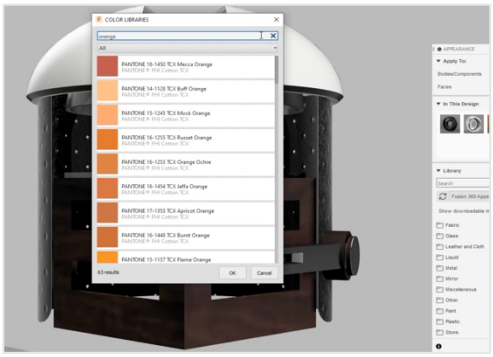
16. Select **Clear Acrylic** on the transparent lens in the helmet's viewport.



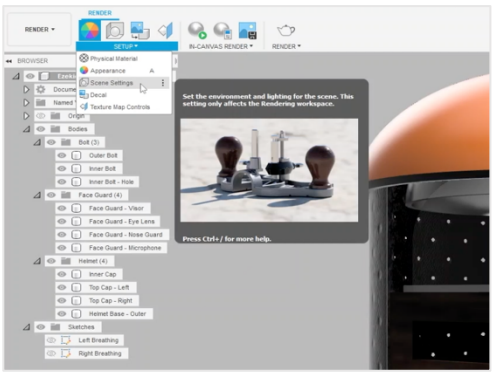
17. You can change the color of a few materials now using the **In This Design** folder. Change the ABS (Plastic) components that were white by default. First, select the **Appearance** from the Setup panel. Hover over the ABS (White) material, right-click and select Edit, and then select **Color Libraries** in the pop-up window.



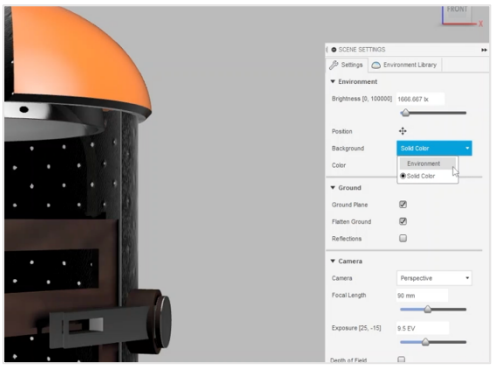
18. In the search bar, type in **Orange**. This will apply the selected orange hue onto all the previously **white ABS** materials. If you'd like, repeat this process with the other materials in **In This Design** to customize your design.



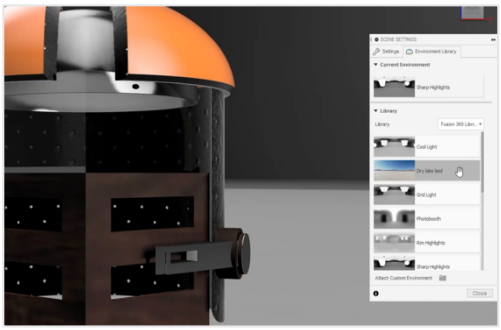
19. Now that you're render is complete, it's time to set your scene! To begin, select **Scene Settings** from the **Set Up** panel. A **Scene Settings** dialog appears, showcasing various options between two tabs: **Settings** and **Environment Library**.



20. In the **Settings** tab, under **Environment**, select **Background** and change the setting from Solid Color to **Environment**. This change allows you to see the environment in the background of your model in the Fusion 360 render workspace.

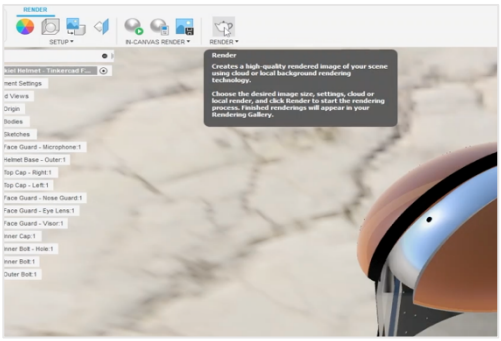


21. Now move back to the **Environment Library** tab and select **Desert** or **Dry lake bed** as the environment. (Note that you may have to download an environment if it's not part of the pre-downloaded environments in Fusion 360.)

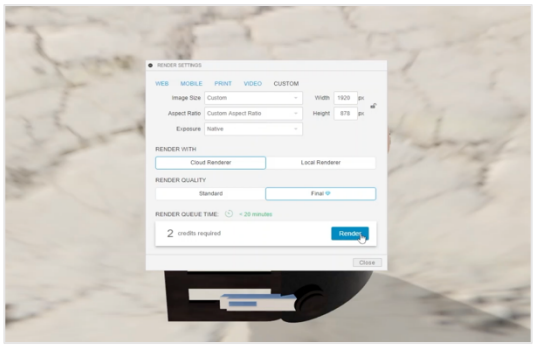


22. Now it's time to transform your canvas into a photorealistic image. The way you position and light your scene is how it will appear in your render. Therefore, you need to set up your canvas for the ideal render. In the Navigation Bar at the bottom of the canvas, use the Rotate command to transform your model into its desired position, as well as the Zoom command to zoom in and out as desired.

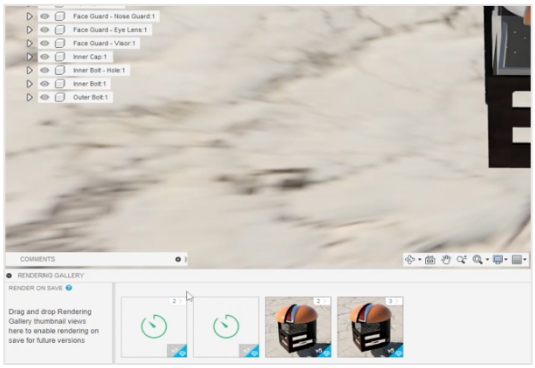
23. Once your model is in the desired position in the scene, you can render your canvas through two methods: in-canvas rendering and cloud rendering. In-canvas rendering is the fastest way to create a photorealistic image of a design. And cloud rendering uses the Internet to product high-quality images of your canvas. Go ahead and select the **Render** command from the **Redner** toolbar in the upper-left side of the window.



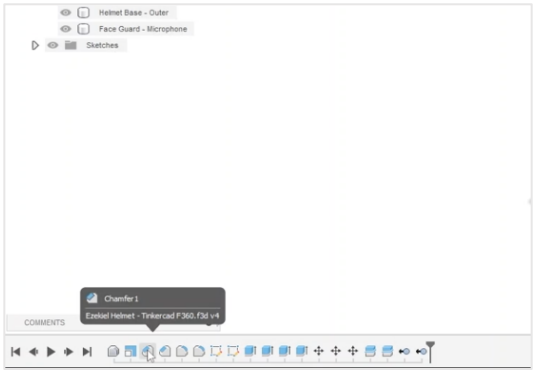
24. A **Render Settings** dialog will appear. Review the settings, make any necessary adjustments, and then click **Render**.



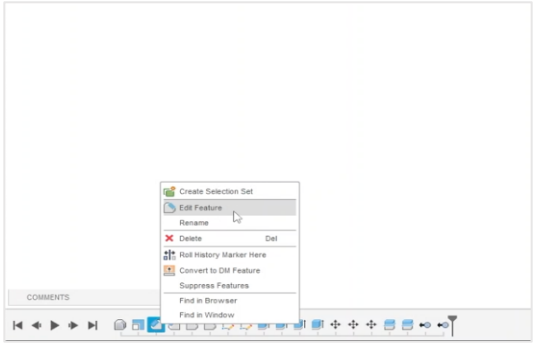
25. It might take a while for the render to complete. You can monitor the rendering process via the **Rendering Gallery**, in the lower-left corner, which shows you thumbnails of the in-progress and complete cloud renderings. To view your full-size renders, just click a thumbnail and download as desired.



26. It's a good idea to create some additional renders with unique scenes and appearances to show off your design.



27. The timeline lets you scroll past your assembly timeline. You can see every command, including the **Chamfer** command, you've completed so far. You can make edits as needed by right-clicking the **Chamfer** command, which opens the dialog. Adjust the Chamfer distance from 5.08 millimeters to 6.35 millimeters.



28. Next, When your edits are complete, you can click **Play** on the Timeline to see how each command has elevated your original design from Tinkercad.