



# F1 in Schools Car Design Intermediate Tutorial

**Abstract:** In this tutorial, you gain knowledge and skills in 3D modeling using Autodesk Inventor to design a 3D model of an F1 in Schools car from scratch. Learn how to assemble models to create a fully completed car, output a detailed engineering drawing from the model, and add dimensions, detail views, and cross sections.

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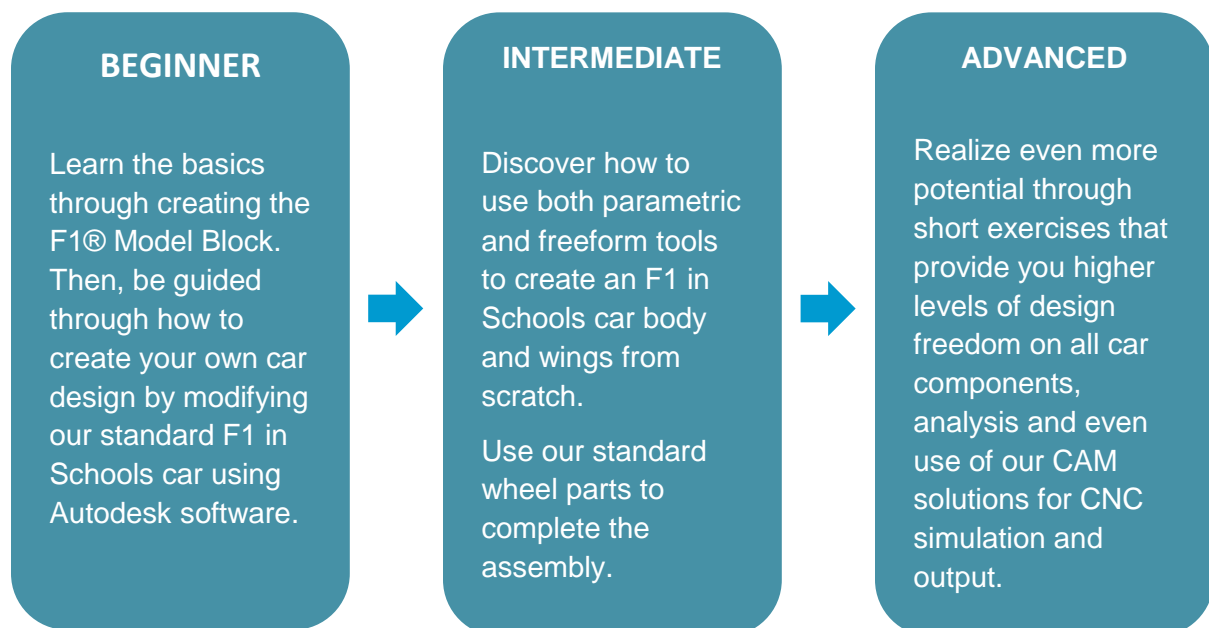
## Introduction

Congratulations on choosing to work through the Autodesk Inventor F1 in Schools™ Car Design tutorials. Ideal for both first time F1 in Schools car designers and more seasoned car designers, these Autodesk Inventor tutorials are designed to get you up and running with the software quickly and effectively so you can have a car ready for manufacturing in record time. Start designing tomorrow's fastest F1 in Schools car the fastest way now!

With these tutorials, you will:

- Get effective results faster. Use the beginner tutorials to learn the software basics. By simply modifying an existing car design, produce your own unique styling ready for manufacturing quickly and successfully.
- Understand the F1 in Schools car body manufacturing process and improve your designs accordingly. Design your car body ensuring it is suitable for CNC manufacturing.
- Save time by having access to a library of standard F1 in Schools car accessory part files, including the standard wheels, axle system, and tether guides.
- Get tips for how to ensure your car body fits the dimensions of the official F1 Model Block.
- Design your car so it has separate front and rear wing parts, fully integrated with the body design, prepared ready to export for manufacturing on a 3D printer.
- Be reminded about F1 in Schools design regulations and how to check compliance as you are designing in 3D
- Extend your skills through quick tips and exercises in our advanced tutorials. Learn wheel design and other features, such as design mass calculation, designing in 3D from a hand sketch, and CAM using Autodesk solutions.
- Use our new Autodesk Flow Design tutorial to analyze the aerodynamic performance of your design in 3D, and make tweaks during your design development stages.
- Be guided in the use of Autodesk Showcase to create stunning photorealistic renders of your final car assembly.

## Tutorial Work Flow:



## Getting Started with Autodesk Inventor

Autodesk Inventor is an application for creating 3D digital prototypes used in the design, visualization, and simulation of products. As a student or educator, you can obtain the Autodesk Inventor software at [www.autodesk.com/education](http://www.autodesk.com/education).

## Starting Autodesk Inventor

To review the Autodesk Inventor user interface, refer to the Beginner tutorial or click on the following link:

[Autodesk Inventor 2016 Tutorial User Interface](#)

## Datasets

All project files that are required for the tutorial are provided in a dataset .zip file. Download the dataset file and extract the files to your computer. Do not modify the file structure of the dataset files.

## Video Tutorials

Download the video tutorials supporting this tutorial. The video tutorials offer the same step-by-step software instruction for learners that prefer guidance through video format.

## F1 in Schools Rules and Technical Regulations

It is extremely important that you design your car to comply with the current F1 in Schools rules and regulations. Each country has slightly different specifications that may change from year to year. It is critical that you download and review your country's F1 in Schools Rules and Regulations documentation and design your car to the outlined specifications. Activities in this tutorial leverage F1 in Schools rules and regulations for 2015-2016 and may not apply to your country.

**Important:** Confirm the F1 in Schools rules and regulations for your country and competition before getting started. Go to your local F1 website and download the F1 in Schools rules and regulations documentation <http://www.f1inschools.com/international-sites/>.

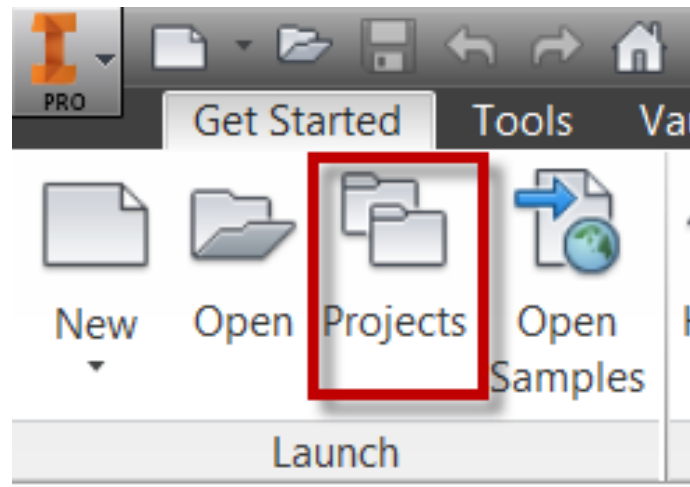
## Activity 1: Model the F1 Model Block

In this section of the tutorial you learn the basics of Autodesk Inventor by modeling the F1 Model Block. For video instructional support, download the step-by-step video tutorials for this activity.

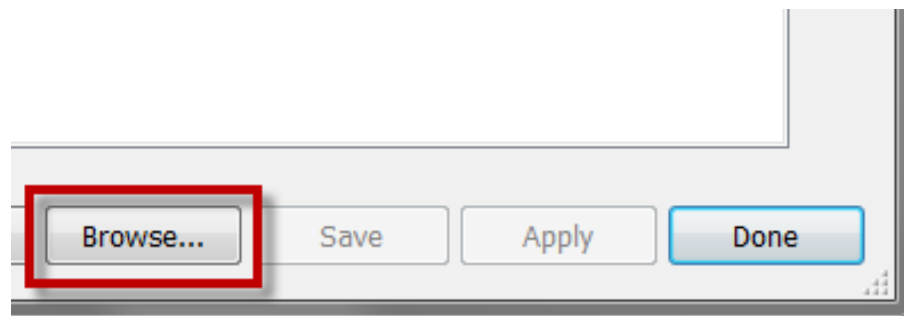
## Step 1: Create a New Part File

Designers typically work on multiple projects. In Autodesk Inventor you use project files to organize and access all files associated with a particular design job. A project file named *F1 in Schools\_Beginner.ipj* is provided for this tutorial.

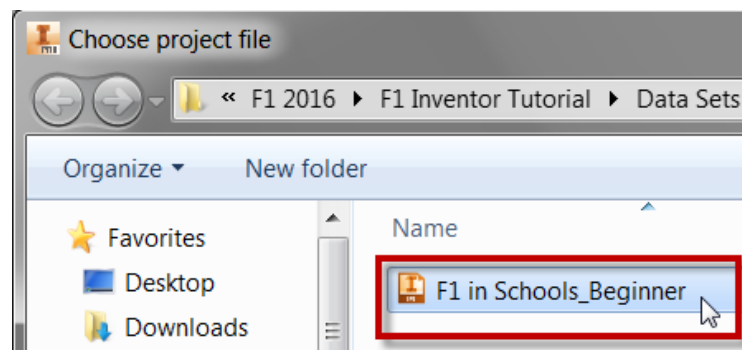
1. Start Autodesk Inventor.
2. On the Launch panel, click Projects.



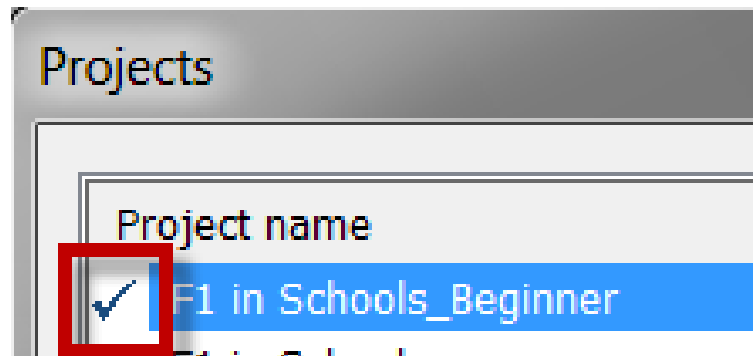
3. On the Projects dialog box, click Browse.



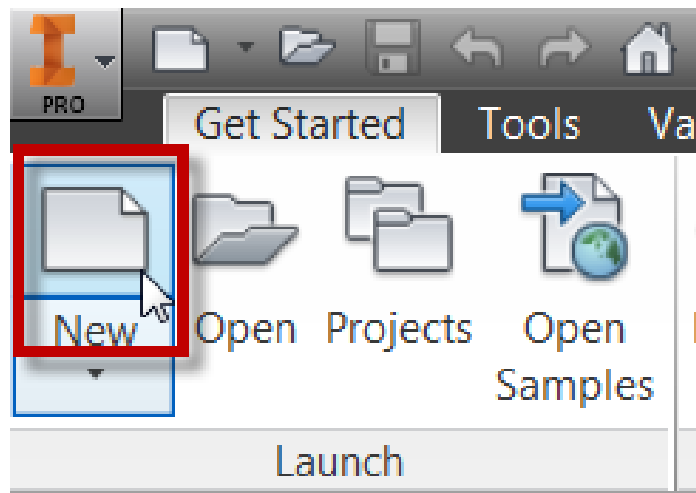
4. Navigate to the location of the project file. Select F1 in Schools\_Beginner then click Open.



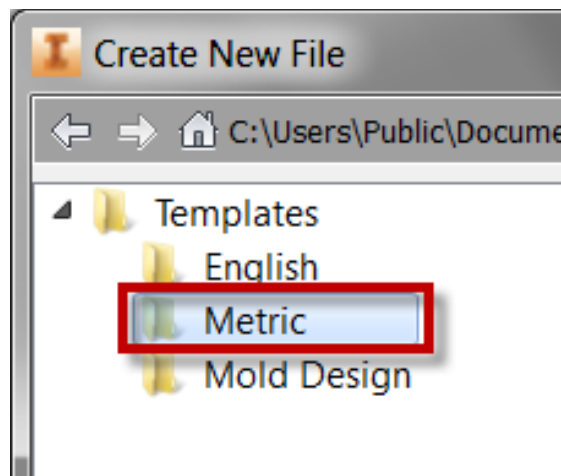
5. Review the Projects dialog box. The active project is indicated with a check mark.



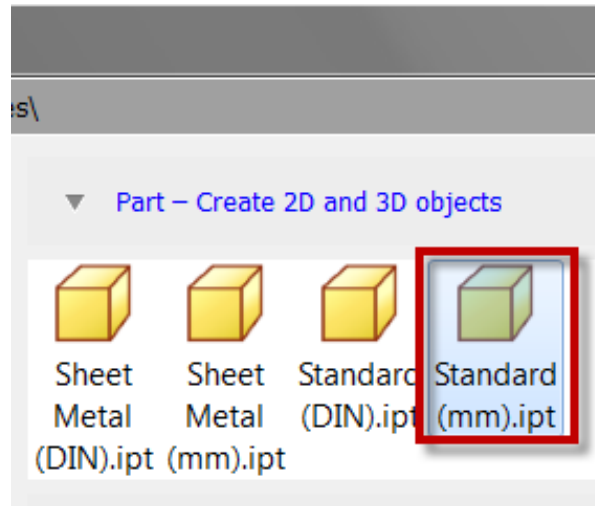
6. Click Done.
7. On the Launch panel, click New.



8. On the Create New File dialog box, under Templates, select Metric.



9. Under Part, select Standard (mm).ipt.



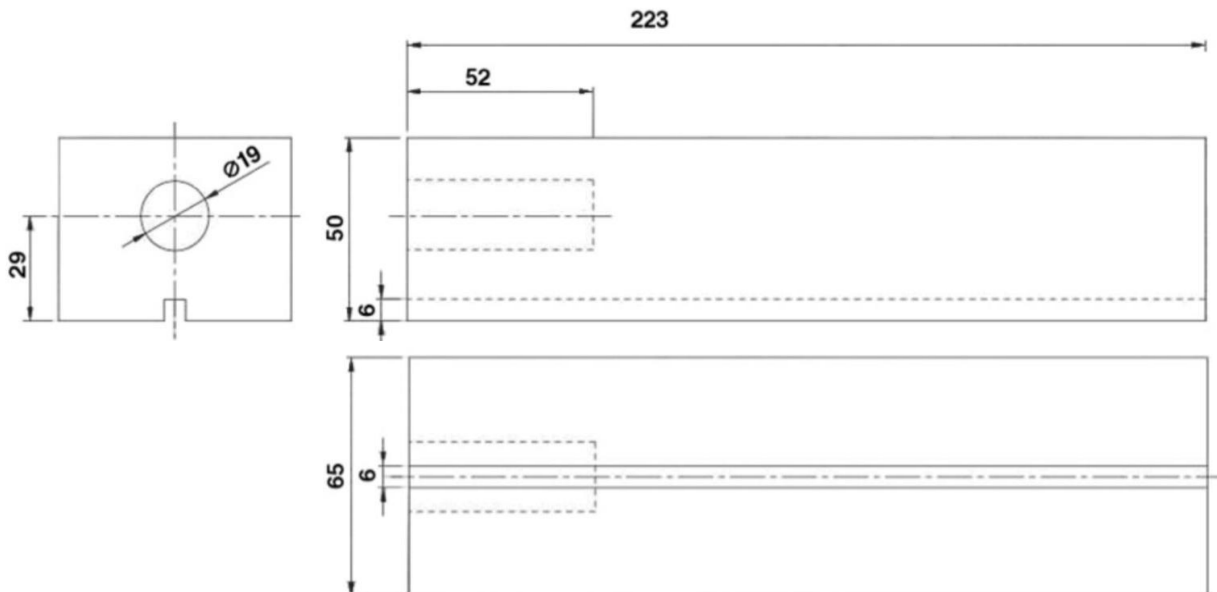
**Note:** A part file has an IPT file extension. This type of file contains a single part.

10. Click Create.

**Note:** F1 in Schools is an international event and uses the metric system for all engineering requirements.

## Step 2: Sketch the Profile of the F1 Block

The first step in modeling the F1 block is to review the dimensions of the official F1 Model Block as stated in the technical regulations. This is the block of medium density modeling foam that all F1 in Schools car bodies are CNC machined from.





To model the F1 Model Block, create a sketch to define the feature profile. For this block you will start by creating a rectangle. The rectangle dimension as stated in the technical regulations is **223 x 65**.

**Tip:** Before starting this activity, review the video on the ViewCube and how to use the ViewCube to view your design.

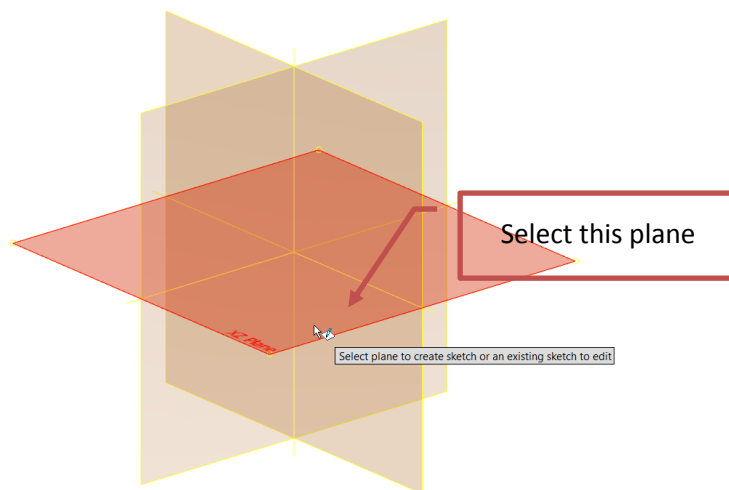
Ctrl+Click on the following link:

[Autodesk Inventor 2016 Tutorial: View Navigation](#)

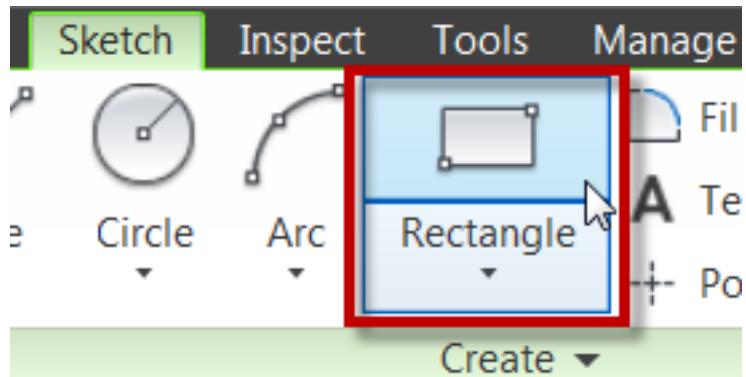
1. On the Sketch panel, click Start 2D Sketch.



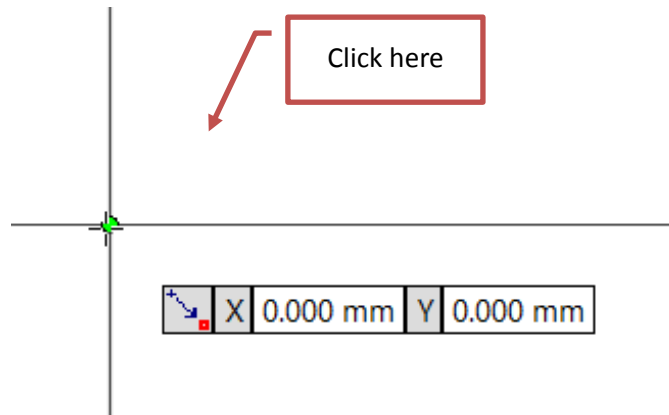
2. Select the XZ Plane.



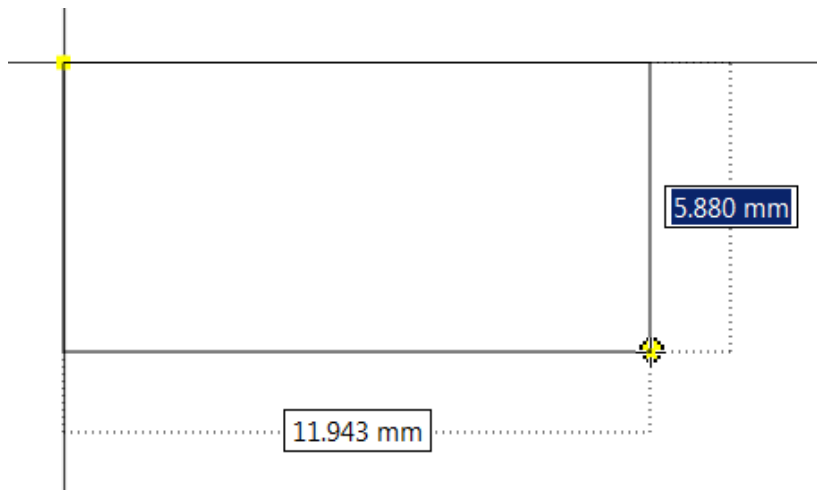
3. On the Create panel, click Rectangle.



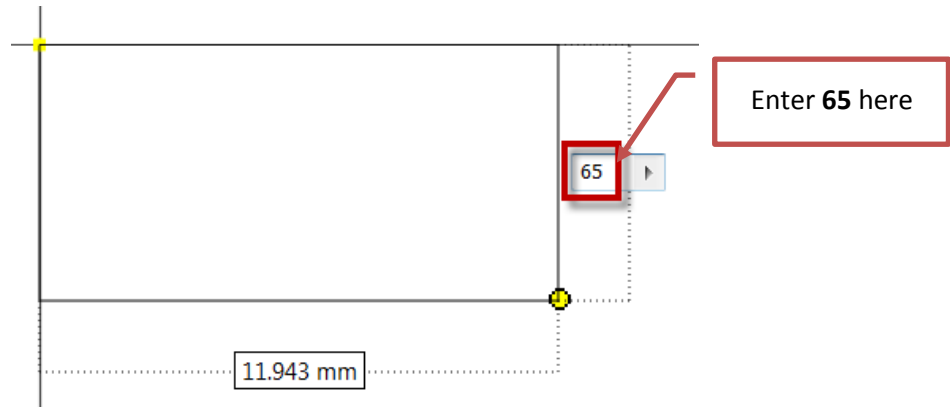
4. Click on the origin of the sketch.



5. Drag the cursor down and to the right.



6. Enter **65**.



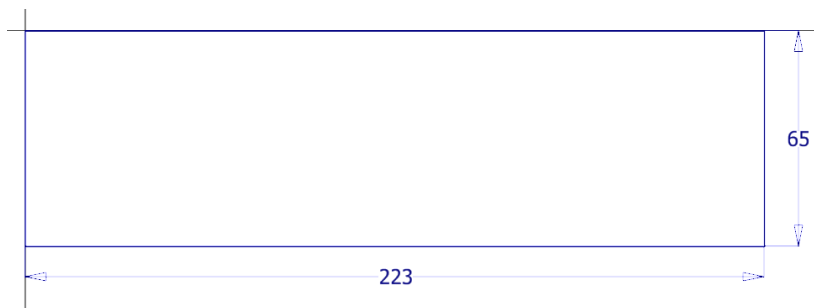
7. Press Tab then enter **223**.

8. Press Enter then right-click and select OK.

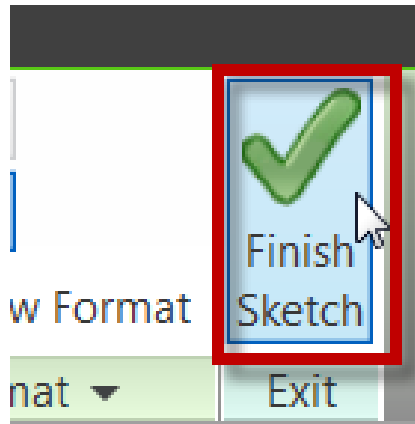
9. On the ViewCube, click Top.



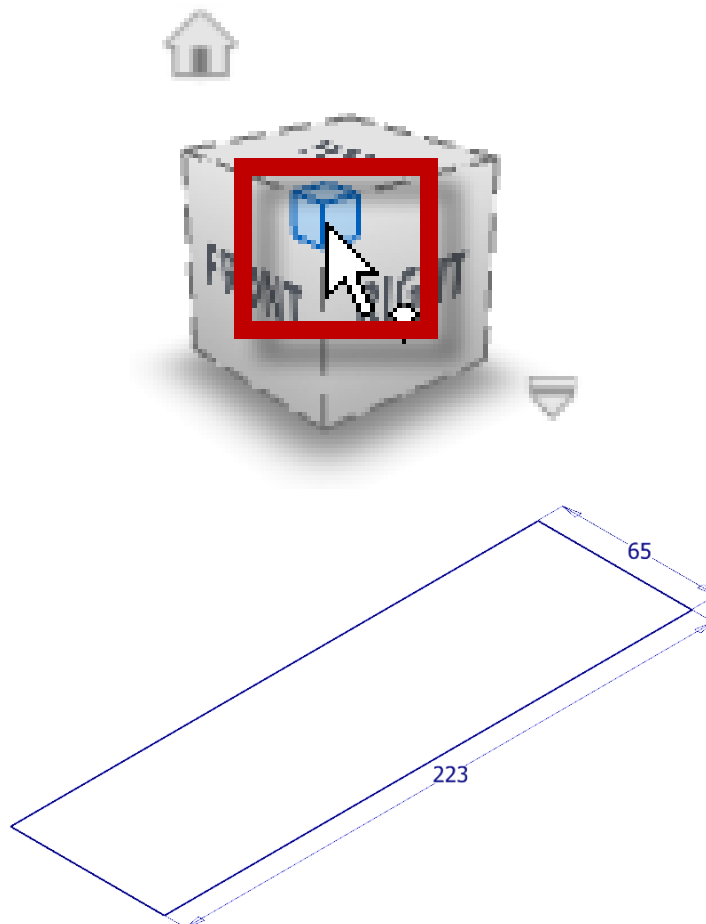
10. Review the rectangular sketch. The length is **223** and width is **65**.



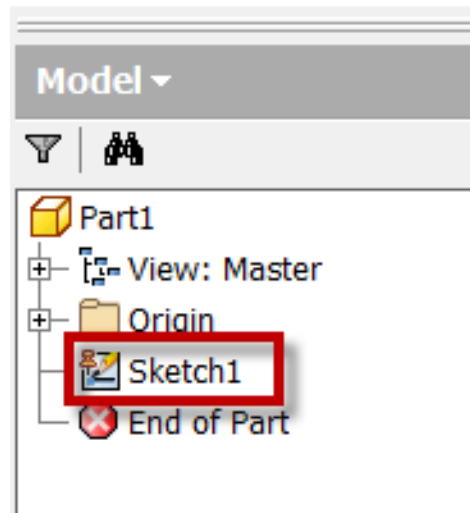
11. On the Exit panel, click Finish Sketch.



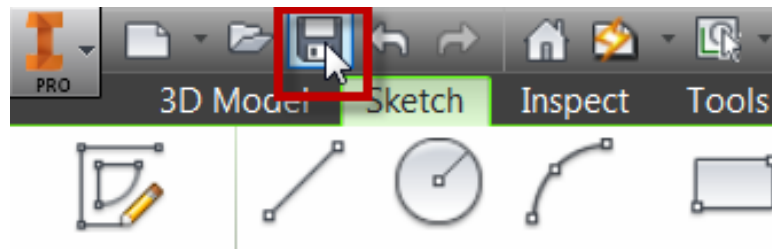
12. On the ViewCube, click the corner as shown.



13. Review the browser. The sketch is added to the part history.



14. On the Quick Access Toolbar, click Save.

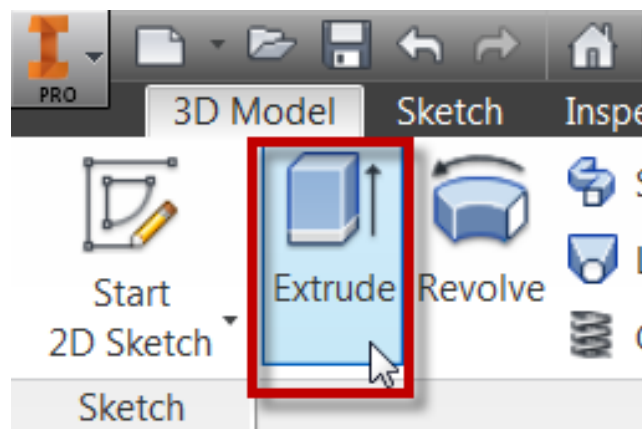


15. Save the file as **my\_f1 Model Block**.

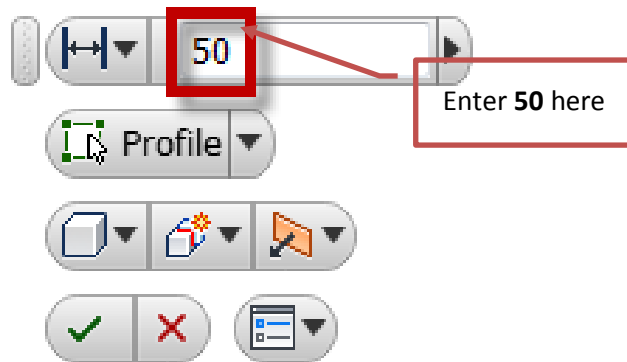
### Step 3: Create the F1 Model Block

The height of the block as stated in the technical regulations is **50 mm**.

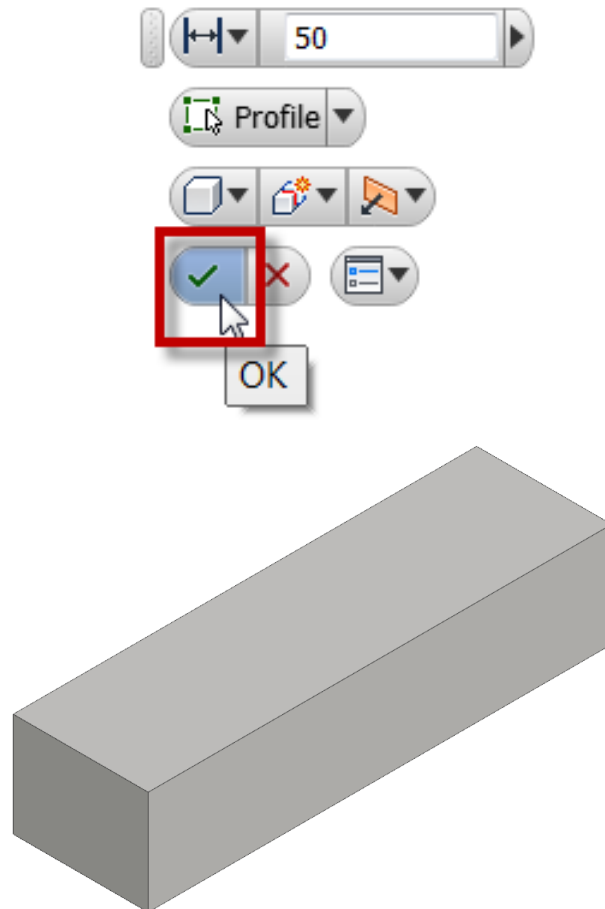
1. On the Create panel, click Extrude.



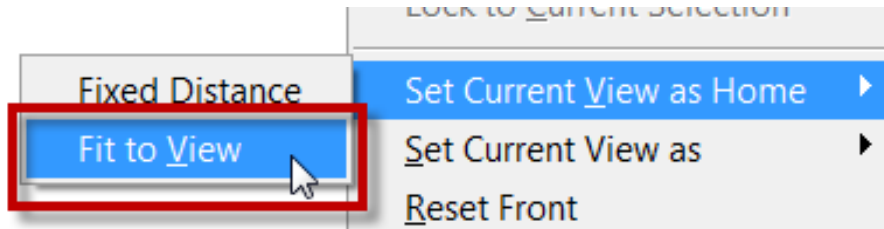
2. For Distance, enter **50**.



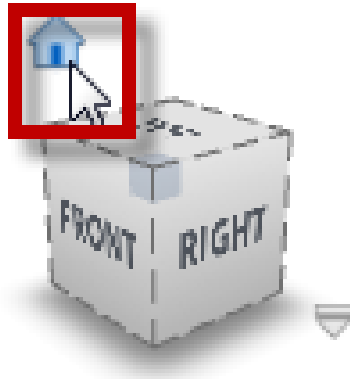
3. Click OK.



4. On the ViewCube, right-click and select Set Current View as Home > Fit to View.

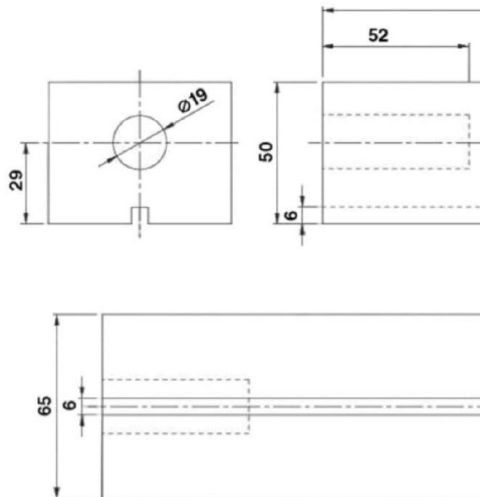


5. On the ViewCube, click Home.

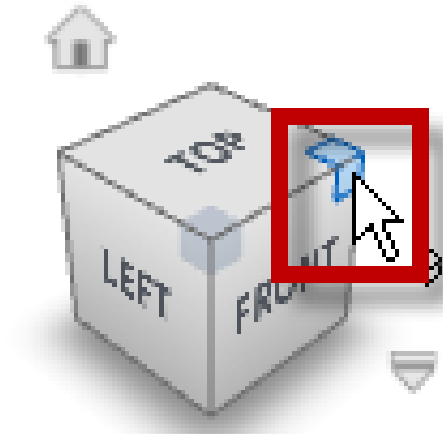


#### Step 4: Create the Cartridge Chamber

The dimensions for the cartridge chamber are documented in the technical regulations as shown.



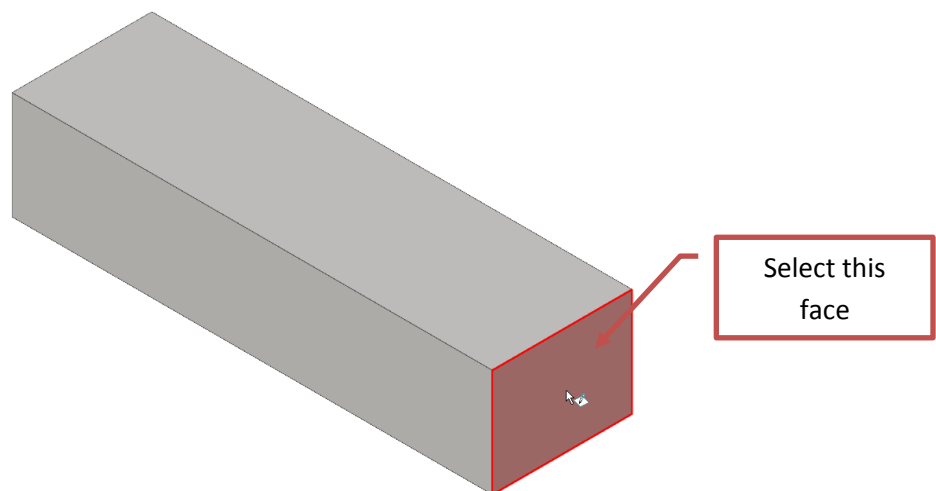
1. On the ViewCube, click the top-right corner.



2. On the Sketch panel, click Start 2D Sketch.

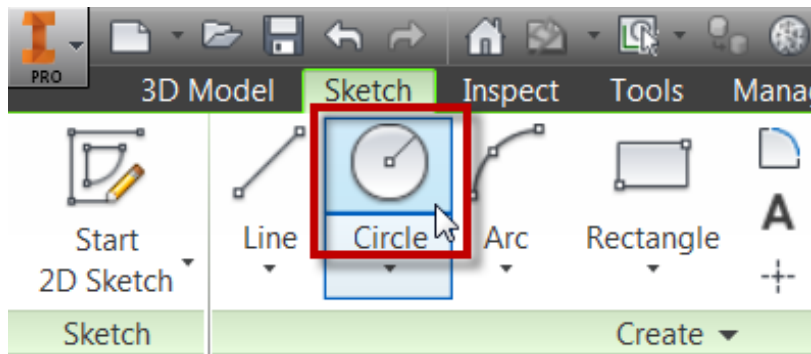


3. Select the end face of the Model Block.

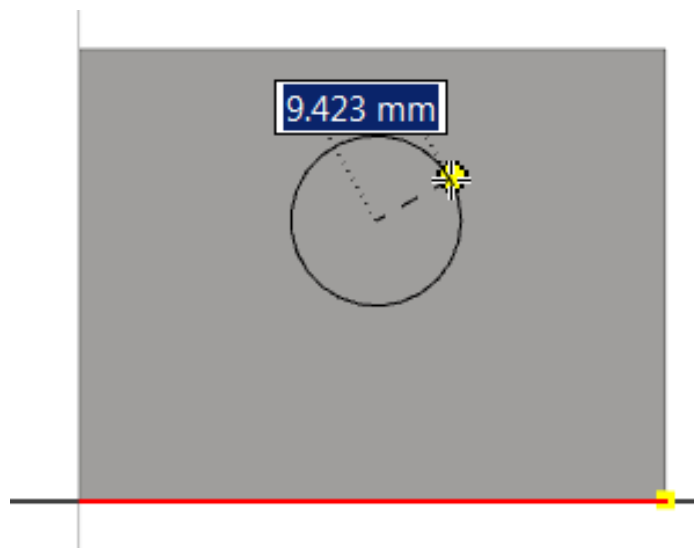




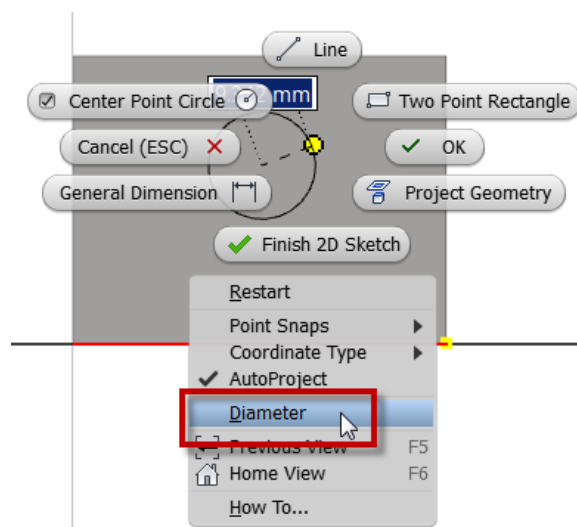
4. On the Create panel, click Circle.



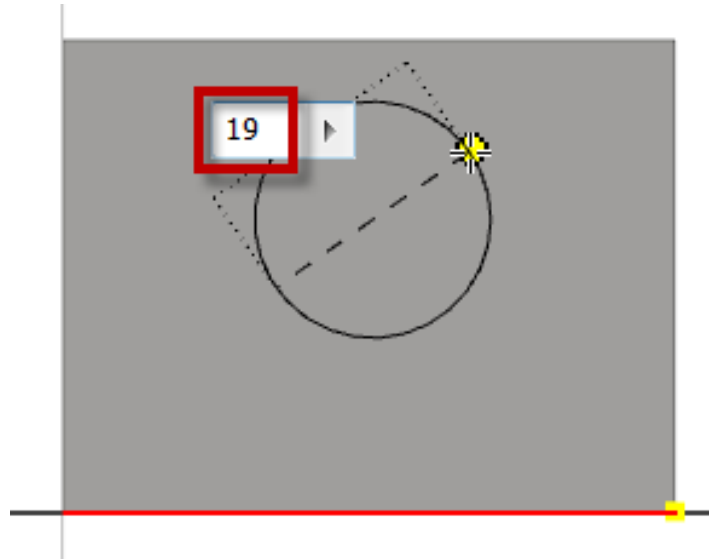
5. Click on the face to set the center point, then drag to preview the circle radius.



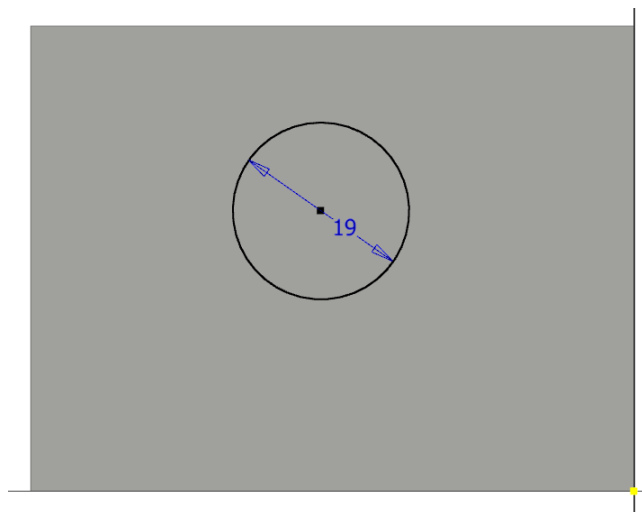
6. Right-click, then select Diameter.



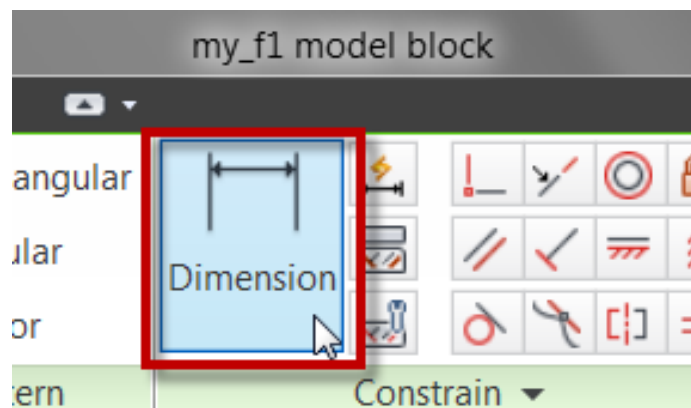
7. Enter **19**.



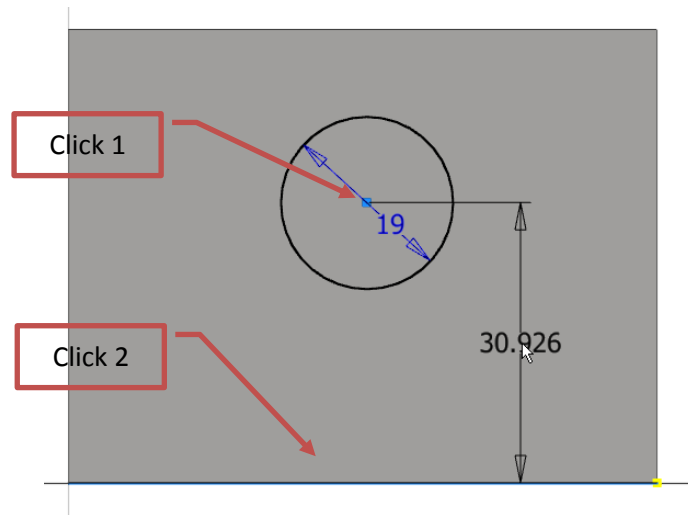
8. Press Enter then right-click and select OK.



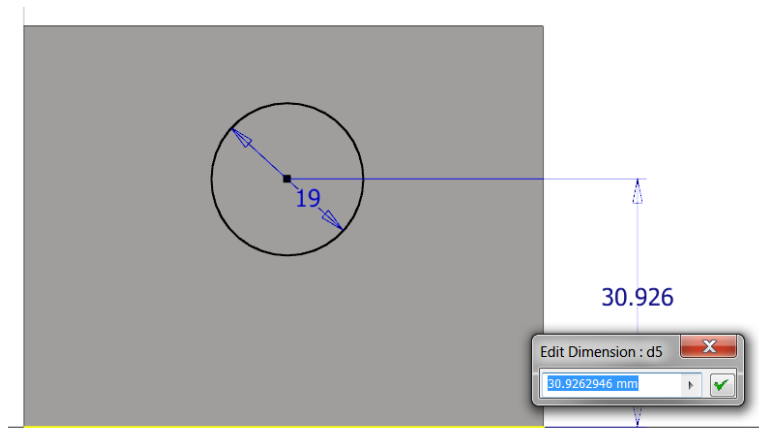
9. On the Constrain panel, click Dimension. When you add a dimension to a sketch it defines the size and location of geometry in the sketch.



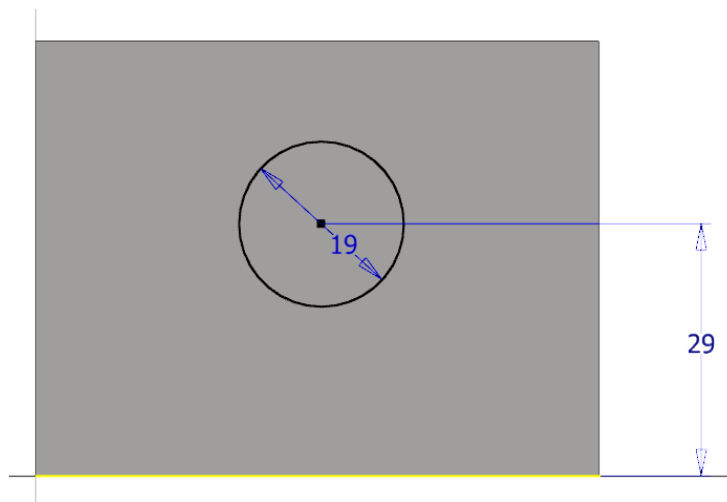
10. Click the center point on the circle (1) then the bottom edge (2).



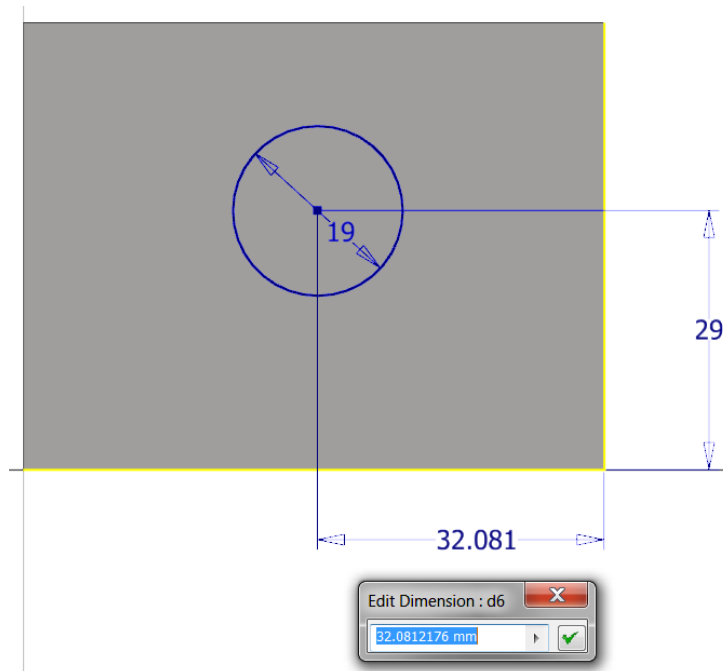
11. Drag the dimension to the right then click to place the dimension



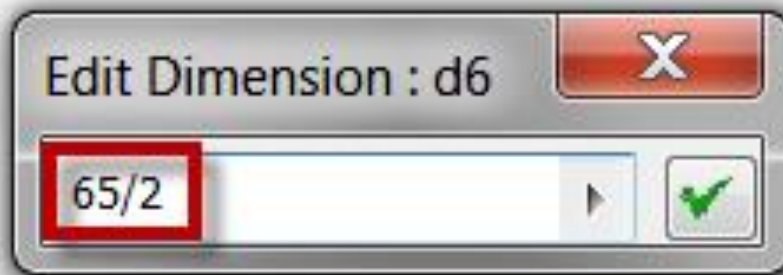
12. Enter **29** then click OK (the check mark). Note the size of the circle changes accordingly.



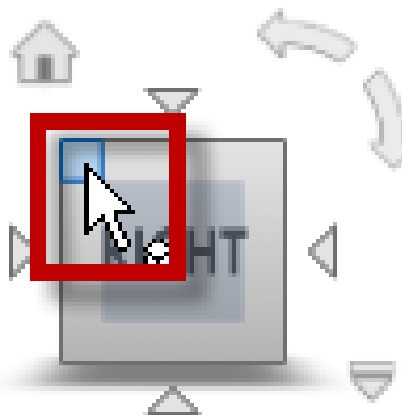
13. Click the center of the circle and the right edge, then drag the dimension below the block and click to place the dimension.



14. Enter **65/2** then click OK. This centers the hole about the 65mm width of the block.



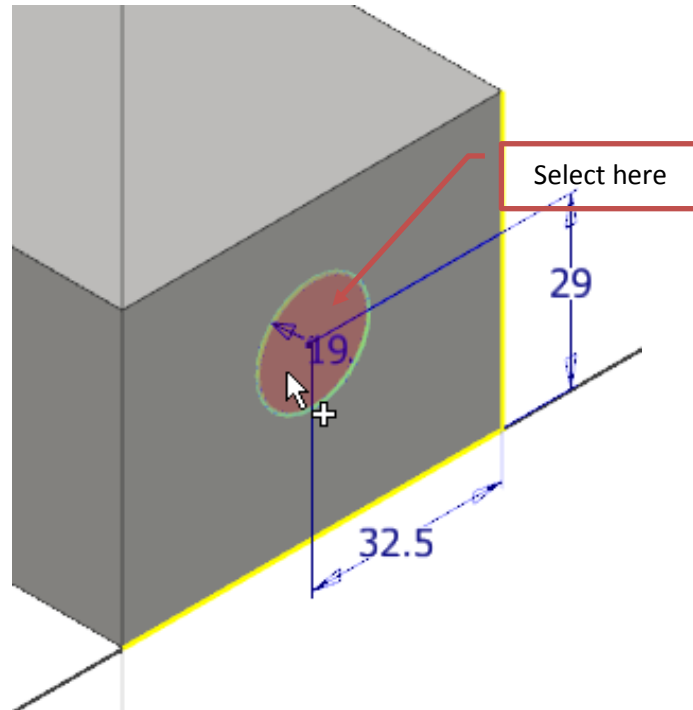
15. On the ViewCube, click the top left corner.



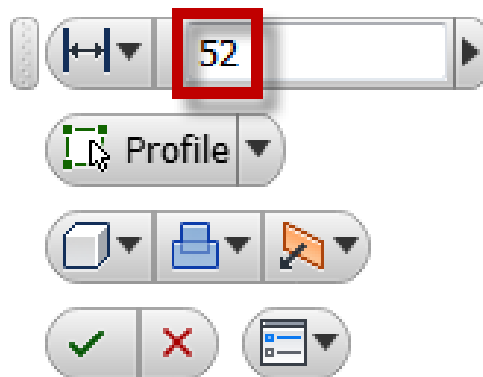
**Note:** To extrude the cartridge chamber you use a keyboard short to start the extrude tool.

16. Press **E**.

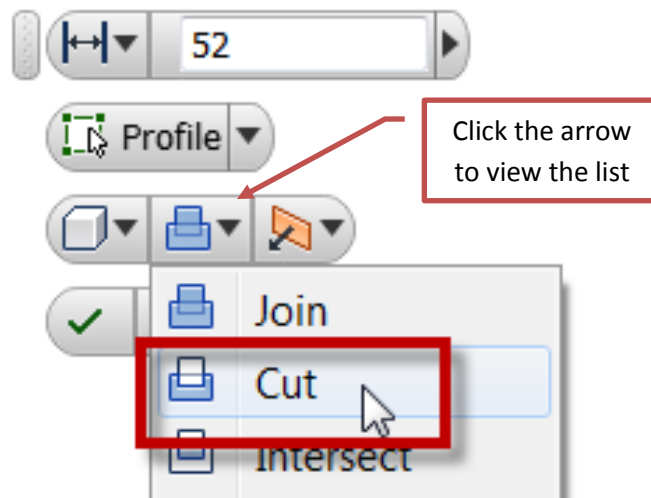
17. Select inside the circle.



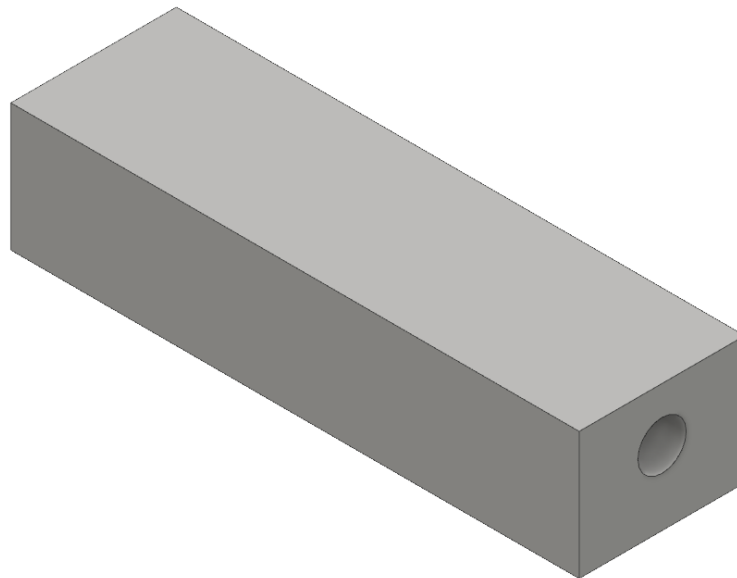
18. For Distance, enter **52**.



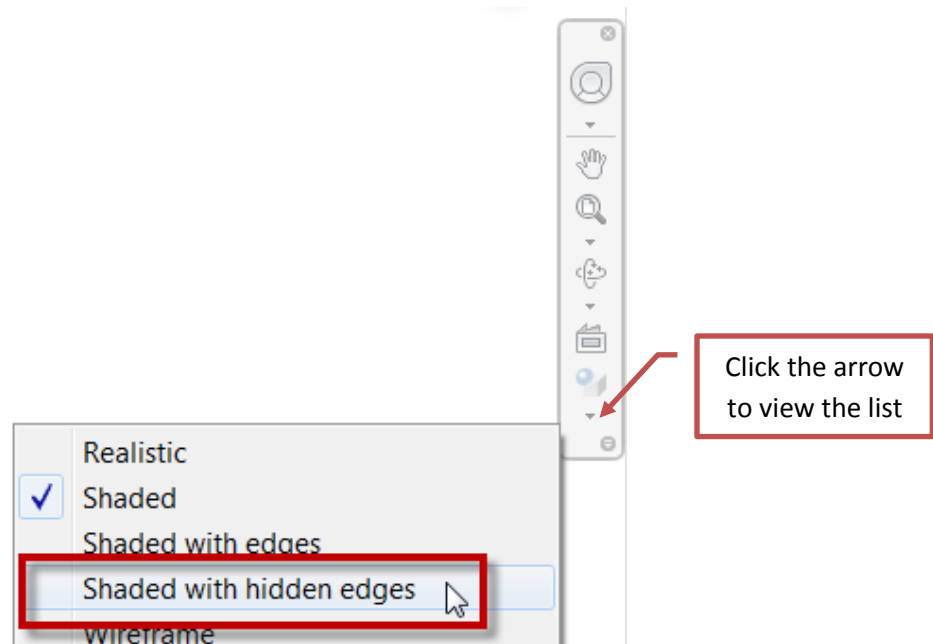
19. From the Operation list, select Cut.



20. Click OK.



21. On the Navigation toolbar, from the list, click Shaded with Hidden Edges.



22. Press and hold the F4 key. Hold down the mouse button then drag to rotate your model.

**Note:** Review other view options on the navigation toolbar such as Wireframe with Hidden Edges

23. On the Navigation toolbar, from the list, click Shaded.

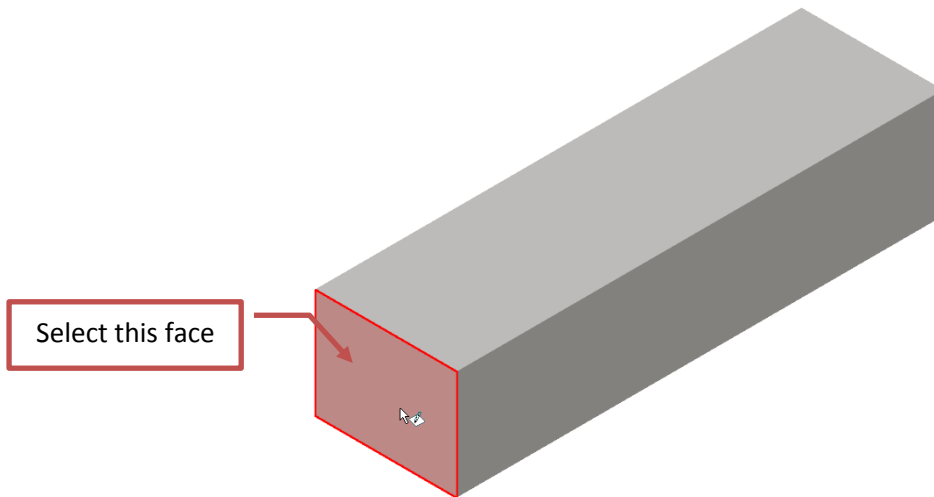
24. Press F6 to return to the Home view.

### Step 5: Create the Tether Line Slot

1. On the Sketch panel, click Start 2D Sketch.

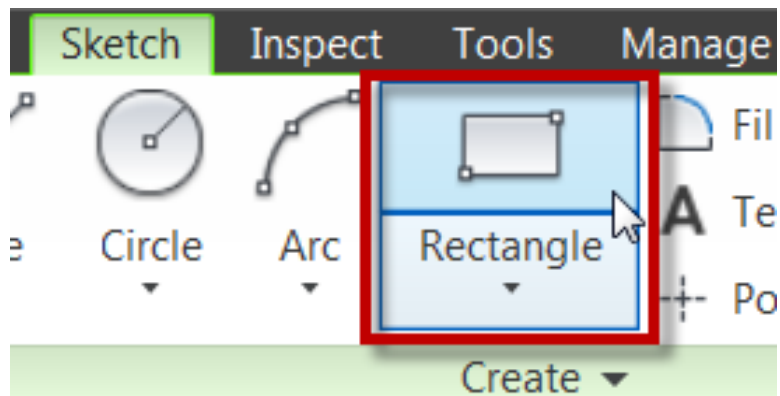


2. Select the end face of the Model Block.

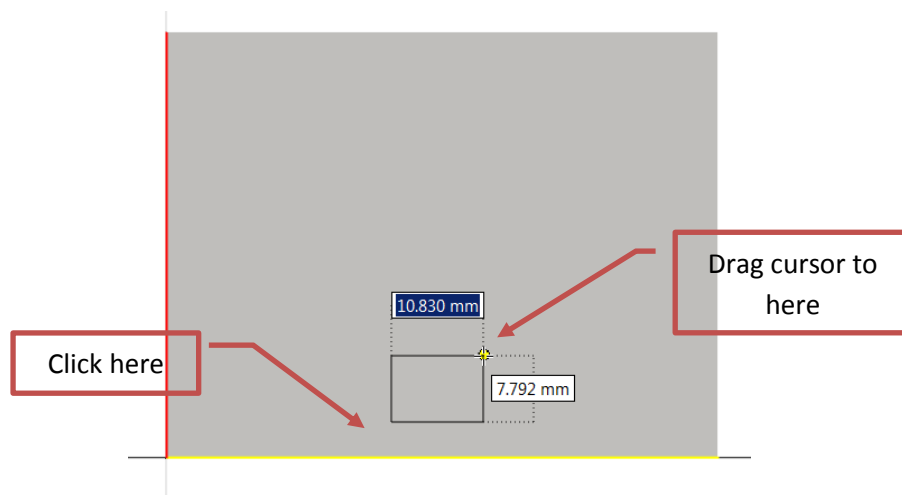


**Note:** The sketch for the tether line slot can be created on either end of the Model Block.

3. On the Create panel, click Rectangle.

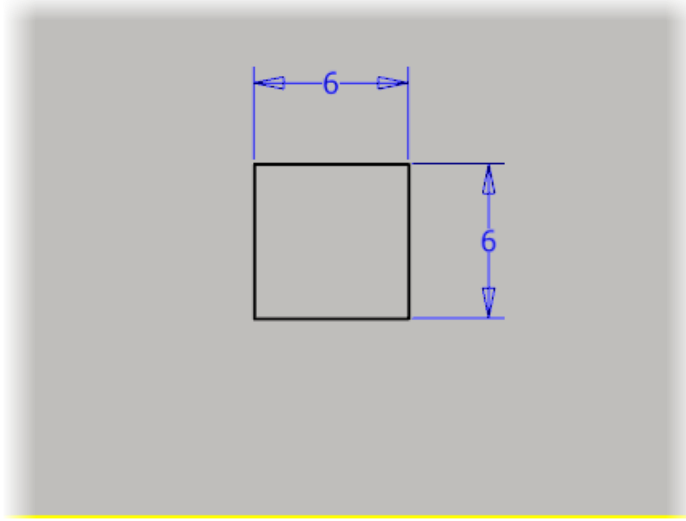


4. Click on the face, then drag up and to the right.

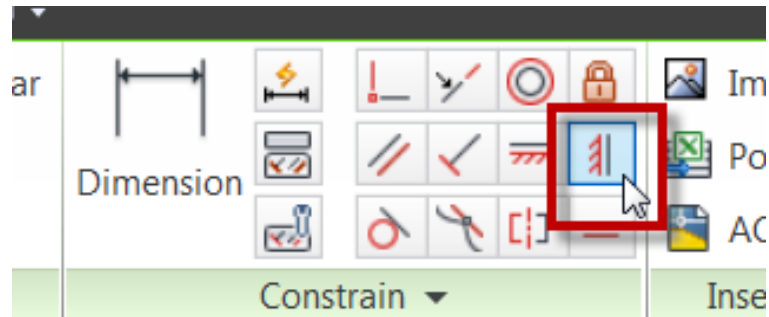




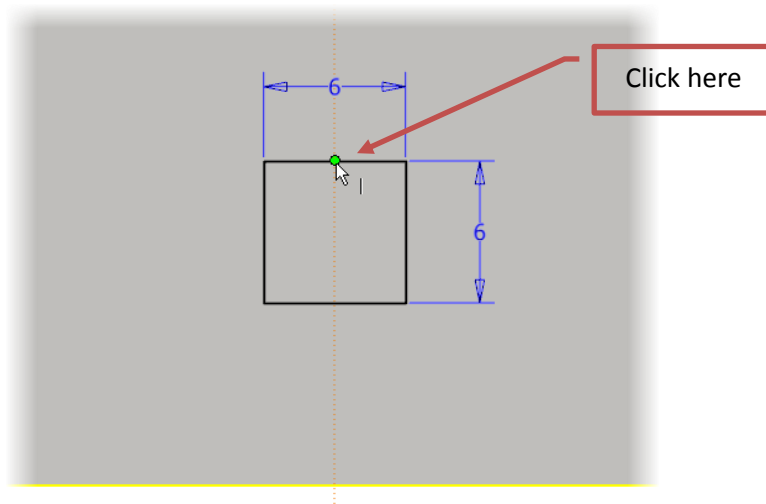
5. Enter **6**, then press Tab and enter **6**. Press Enter, right-click, and then click OK.



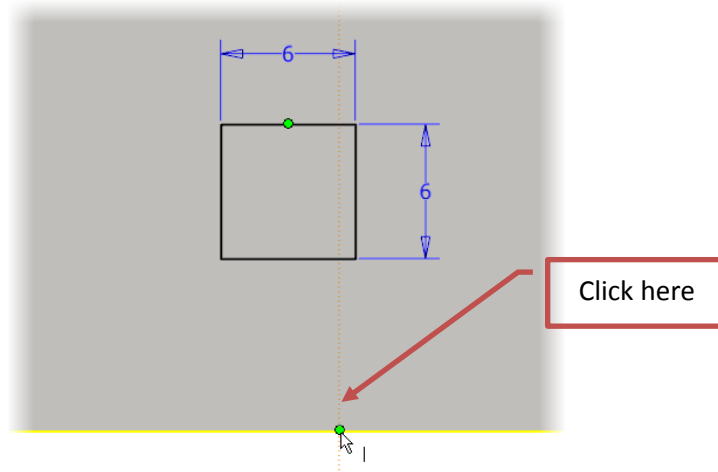
6. On the Constraints panel, click Vertical.



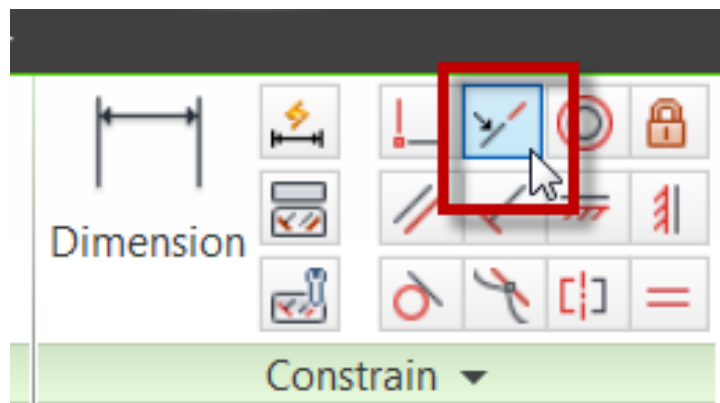
7. Click the midpoint on the upper edge on the rectangle.



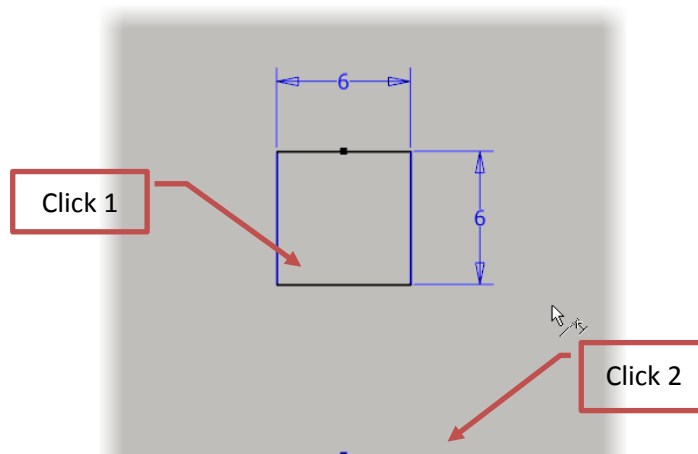
8. Click the midpoint of the lower edge of the block. It is indicated by a large dot.



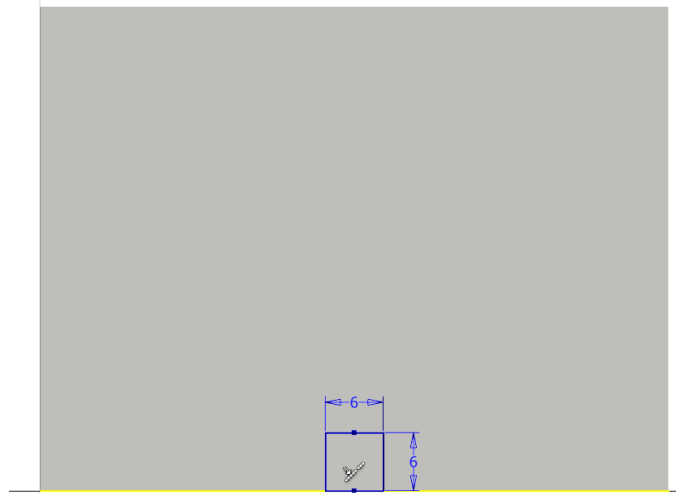
9. On the Constrain panel, click Colinear.



10. Click the bottom edge of the rectangle (1) then the bottom edge of the block (2).

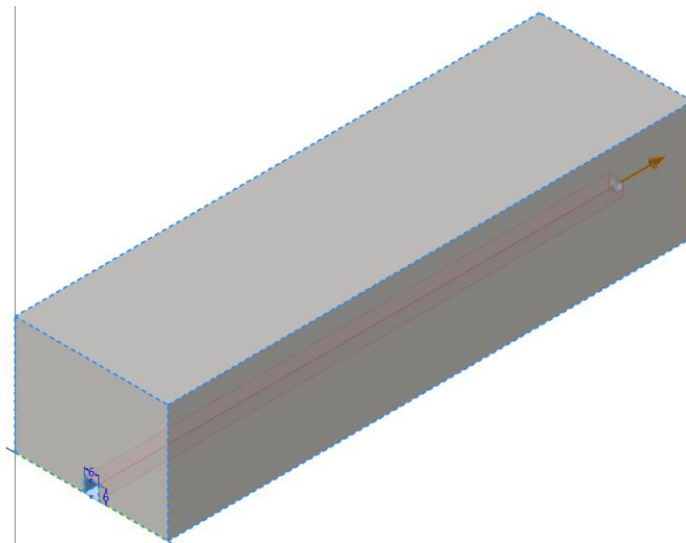
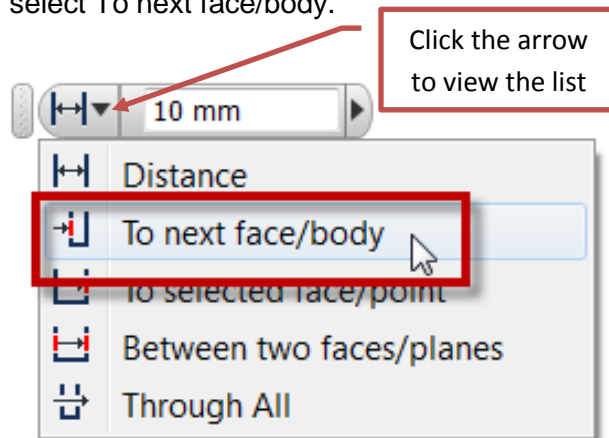


The rectangle is now dimensioned and located as stated in the technical regulations.

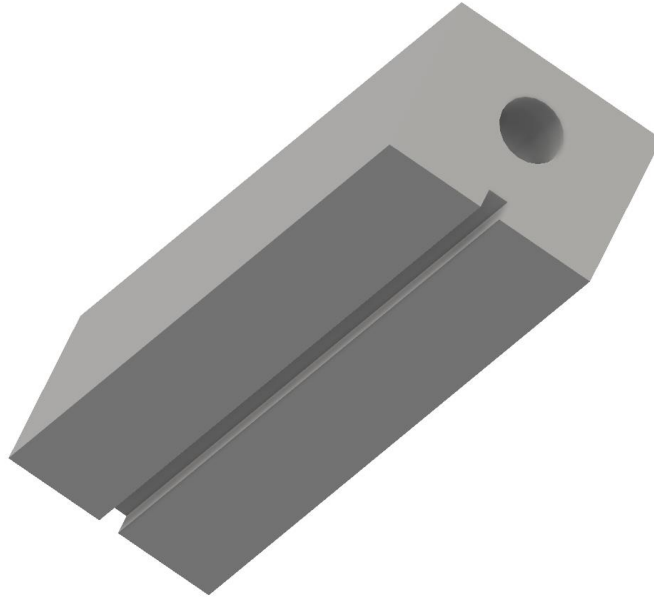


11. Press **E**.

12. On the Extents list, select To next face/body.



13. Click OK.
14. Press and hold the F4 key. Hold down the mouse button then drag to rotate your model to view the tether slot and cartridge chamber.



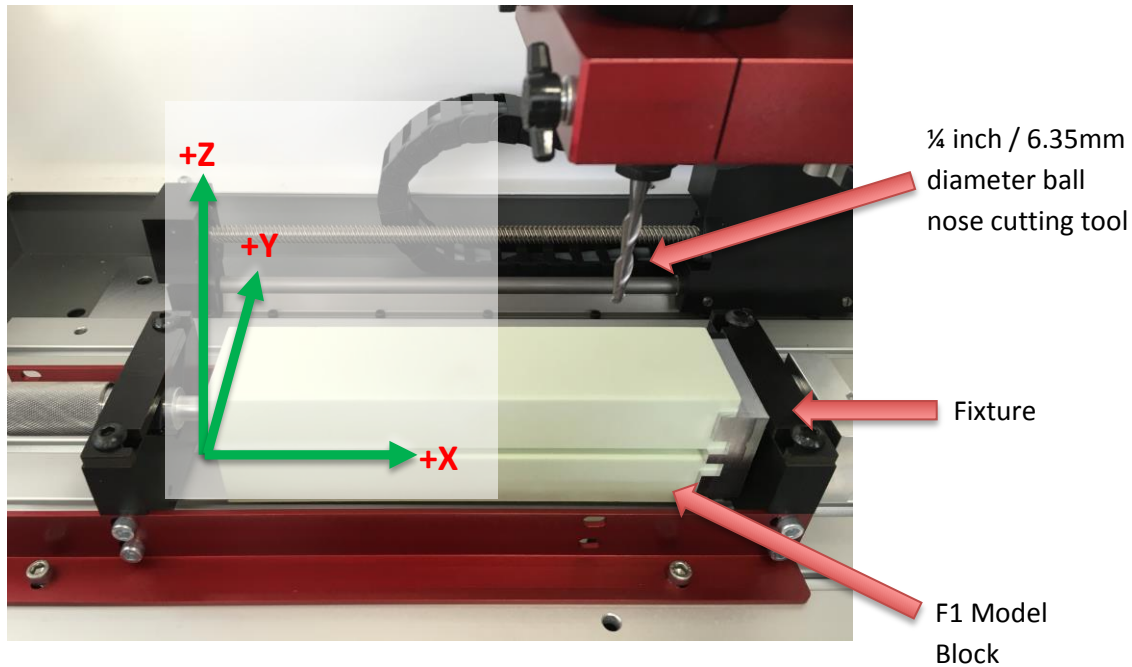
15. Press F6 to return to the Home view.
16. Save the file as **my\_f1 Model Block**.

### Step 6: Design for Manufacturing Considerations

It is very important to have a basic understanding of how the car body will be manufactured in order to create a successful design. In industry, all products are designed with consideration given to any limitations that may exist due to the available or chosen manufacturing methods. An F1 in Schools car is no different. It is important to understand the manufacturing technology and process requirements so that the car designer can create a car body shape that is actually possible for a CNC machining process to manufacture.

CNC (Computer Numeric Control) machining is a very important and widely used manufacturing method in industry. This is why it is compulsory for F1 in Schools teams to manufacture their car bodies using a CNC machine. Many schools are equipped with 3 axis CNC machines—the minimum requirement for car body manufacture. High speed 3 Axis CNC Routers are ideal for making F1 in Schools car bodies. Autodesk F1 in Schools tutorials support manufacturing via a Denford 3 Axis CNC Router.

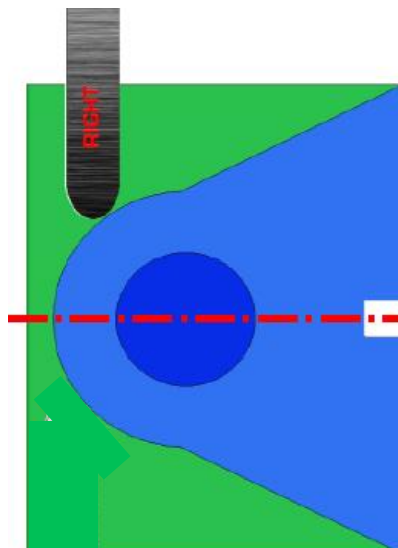
The recommended standard CNC machining method is to machine car bodies using two machining operations: one operation from the left side of the car and one from the right side. For each operation, the F1 Model Block is mounted on its side in the CNC machine via clamping in the special F1 in Schools Machining Fixture. This method provides for a wide range of car profile designs to be machined with a minimum amount of machining operations. The picture below illustrates this.



The green arrows above indicate the 3-axis direction system of the CNC machine. These are the 3 axes of direction that the cutting tool moves in, often simultaneously, in order to remove the excess Model Block material revealing the profile of your F1 in Schools racer.

The standard method of manufacture is to machine the car body using two machining operations, one machining operation on each side of the car. The right side of the car is machined with the Model Block mounted as pictured above, the Model Block is then rotated 180 degrees about the x axis to machine the left side of the car body. The left side machining operation is simply a mirror image of the right side.

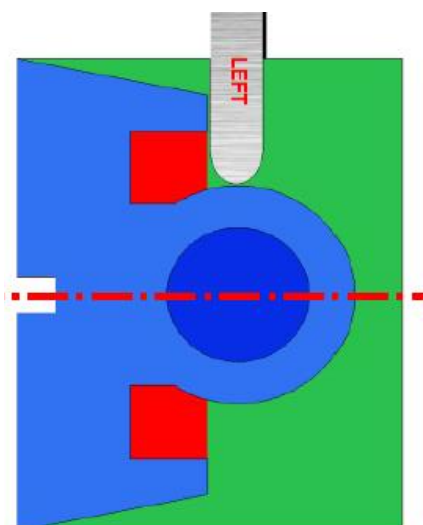
Due to there being only 3 axes or directions of movement that the cutting tool can move in, some shapes or profiles may not be possible to machine. The diagrams below help explain this further



The Model Block (green) and car profile (blue) are shown oriented on their side in the CNC machine. The dark blue circle is the CO2 canister hole in the Model Block.

The cutting tool (shown in grey) can remove all of the material to produce the design (shown in blue) as it can be reached from left and right sides of the car.

The material shaded in red cannot be removed as the tool cannot reach this area from the left and right sides.

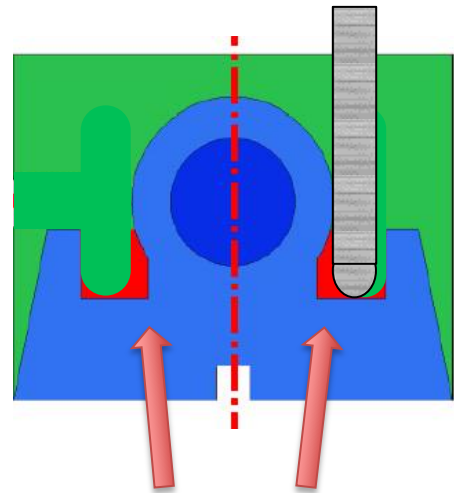


As pictured on the right, it is possible to mount the Model Block in the CNC Router orientated to allow for machining from the car body top. This can then also be inverted to allow for machining of the car body bottom if required. These would be additional machining processes increasing complexity, time, and cost to machine the design. Note that, with this design, there are still areas (shown in red) that the cutter cannot access; these areas are not possible to machine using any orientation.

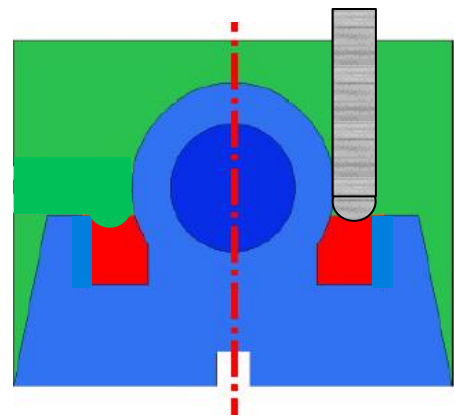
Finally, the size and shape of the cutting tool needs to be considered. The standard machining process for F1 in Schools car bodies uses a ¼ inch (6.35mm) diameter ball nose shaped cutter. Looking at the diagram above you will notice that a small radius/fillet of material (in red) remains where two edges meet at an internal sharp corner. This is due to the ball nose shape of the cutter tip.

The example to the right is a similar body profile to the above example. However, here the slot on the left and right sides of the CO2 canister housing is not as wide. The slot is narrower than ¼ inch/6.35mm and therefore the cutter cannot access and remove the material in red. Any slot or groove features that are narrower than the diameter of the cutter cannot be machined.

Hopefully now you have a better understanding of the CNC machining process used to manufacture F1 in Schools car bodies and the few limitations that this imposes. You should now keep these factors in mind as you unleash your creativity in designing your F1 in Schools racecar bodies.



ABOVE: Material remains in the sharp internal corners (shown in red) due to the ball nose shape of the cutter tip (shown in grey). Model Block material that will be machined away is indicated in green

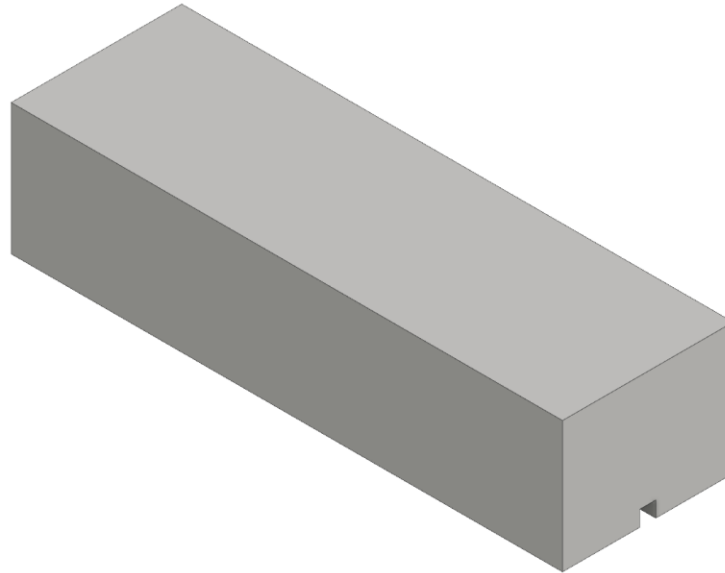


## Activity 2: Adding Axle Holes and Cutting the Block

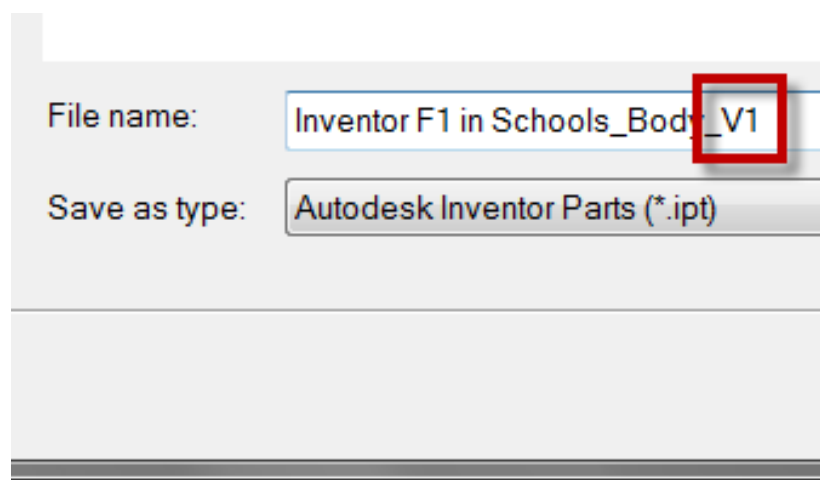
In this activity, you add axle holes for your wheels and modify the block to allow room for the wheels. You will use your F1 in Schools block created in the Beginner tutorial or in the previous activity. For video instructional support, download the step-by-step video tutorials for this activity.

### Step 1: Modifying the Block Length

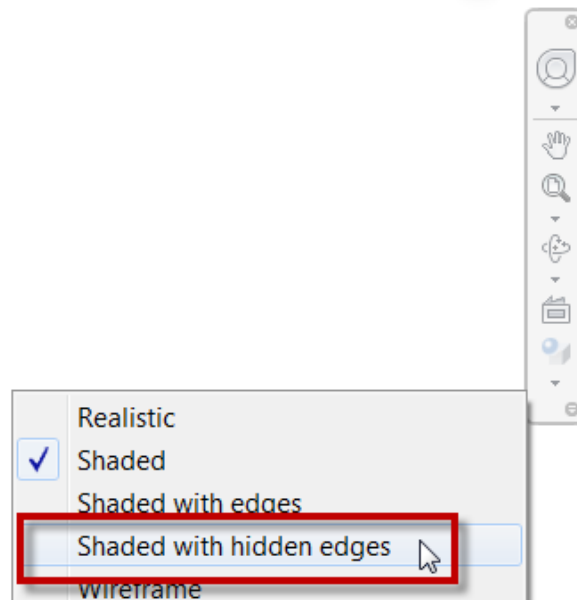
1. Open your Model Block.



2. On the Application menu, click Save As > Save Copy As.
3. For File name, enter a new name for your block. In this example **\_V1** was added to the original filename. The new file is a backup should you need another Model Block.

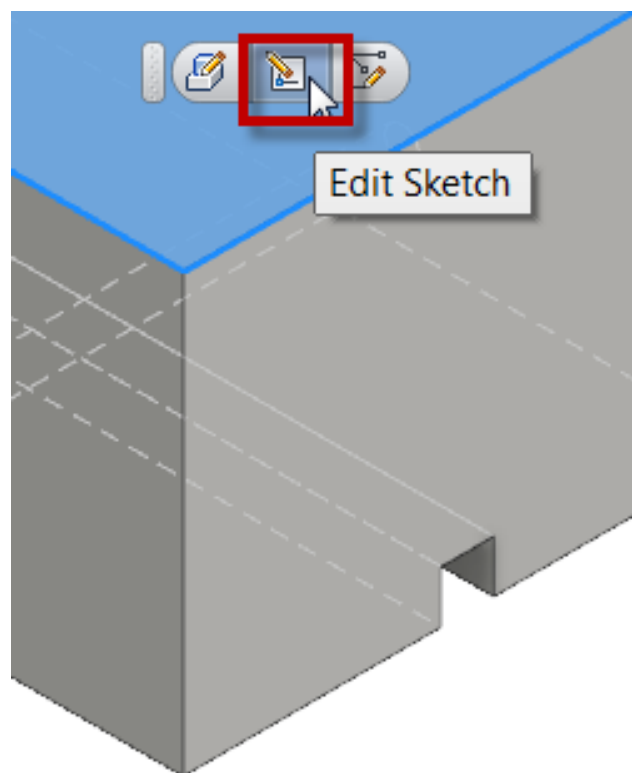


4. On the Navigation toolbar, from the list, click Shaded with Hidden Edges.



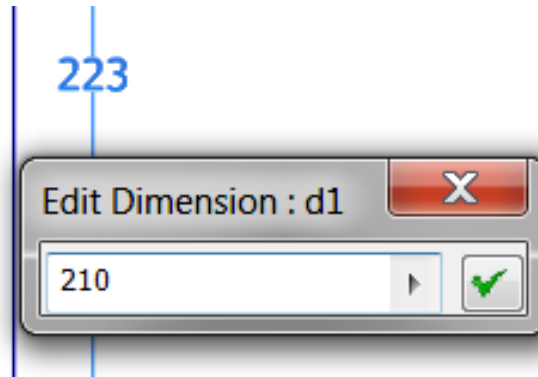
**Note:** The supplied F1 in Schools Block is 223mm in length. The F1 Technical Regulations state the overall car length must be a maximum of 210mm only. Also, this is the maximum size the CNC router can cut due to clamping of the block in the machining fixture.

5. In the graphics window, select the top face as shown, then click Edit Sketch.

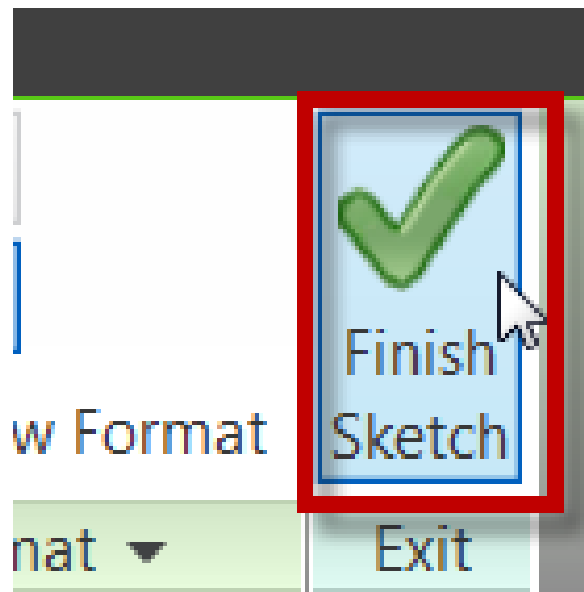




6. Double-click the 233 dimension, then enter **210**.



7. Click OK (the check mark). The size of the model changes accordingly.
8. On the Exit menu, click Finish Sketch.



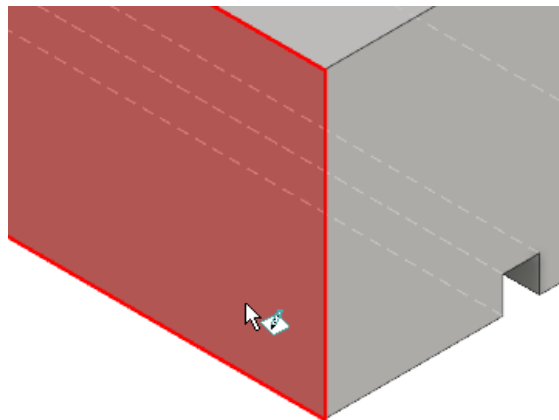
## Step 2: Creating the Axle Holes

To create a hole, you add points to the sketch. These points are the center of the hole features.

1. On the Sketch panel, click Start 2D Sketch.

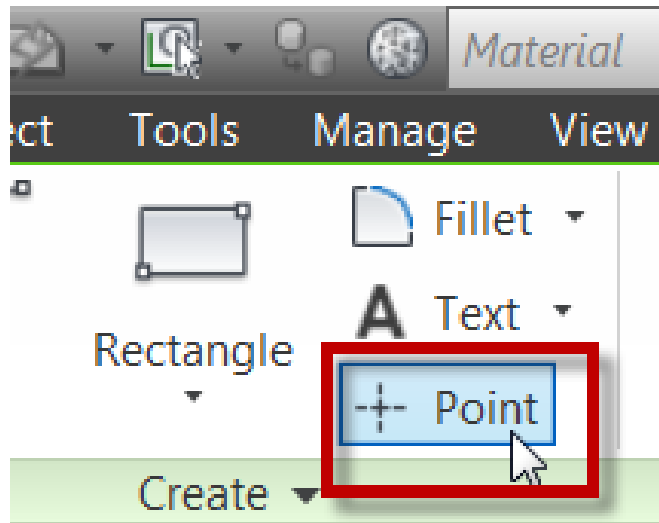


2. Select the front face of the block as shown.

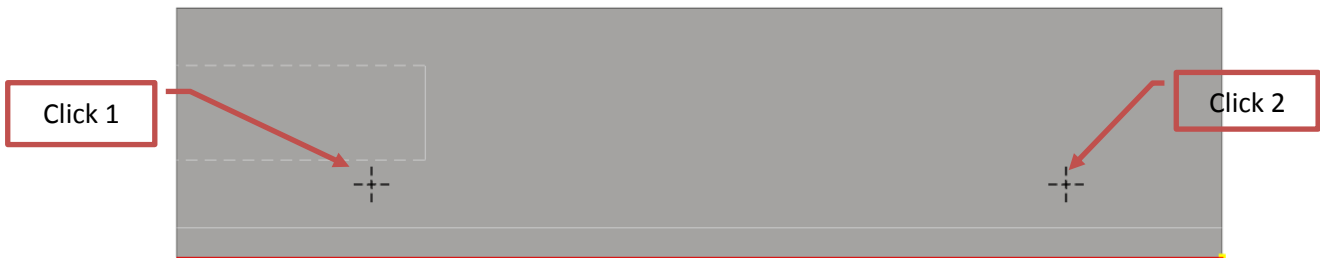


3. On the ViewCube, click Left.

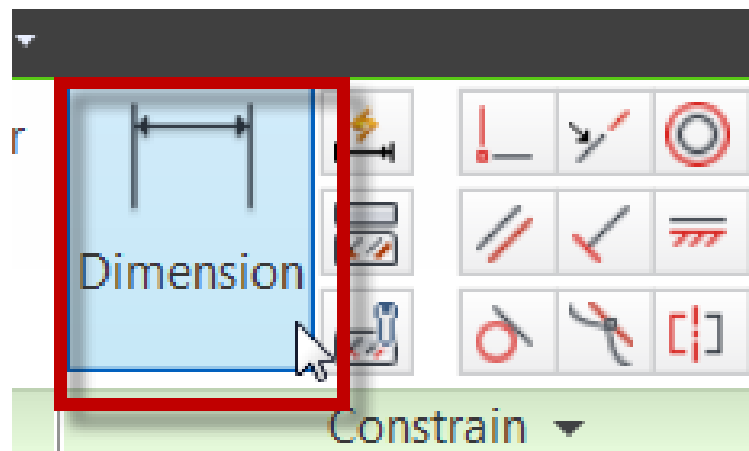
4. On the Create panel, click Point.



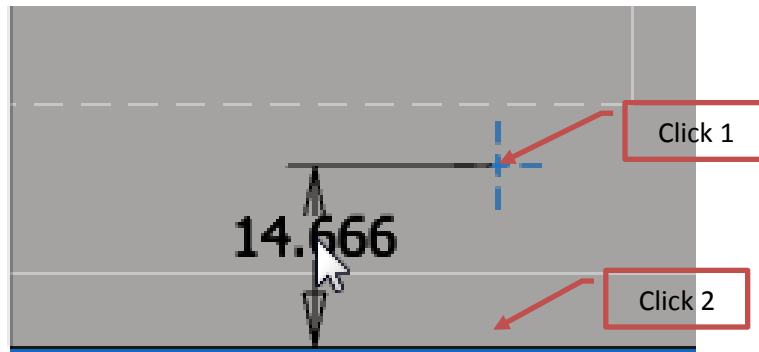
5. Click on the sketch twice to place the points.



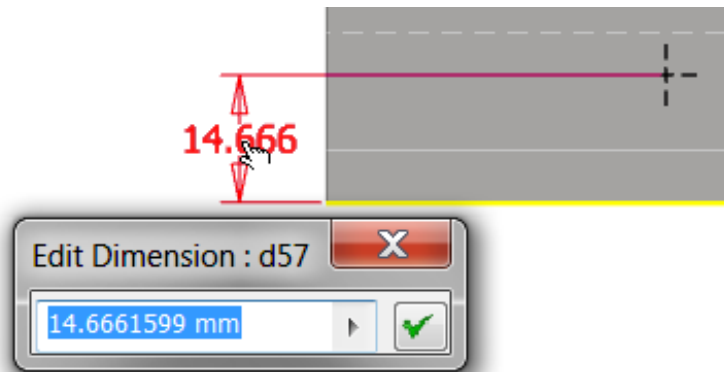
6. On the Constrain panel, click Dimension.



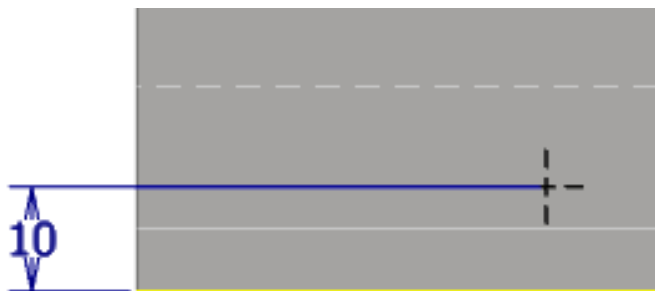
7. Click the point on the left (1), then click the bottom edge of the block (2).



8. Drag the dimension to the left, then click to place the dimension



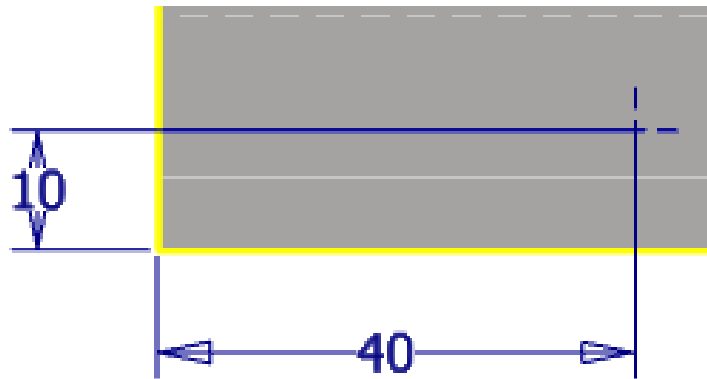
9. Enter **10**, then click OK.



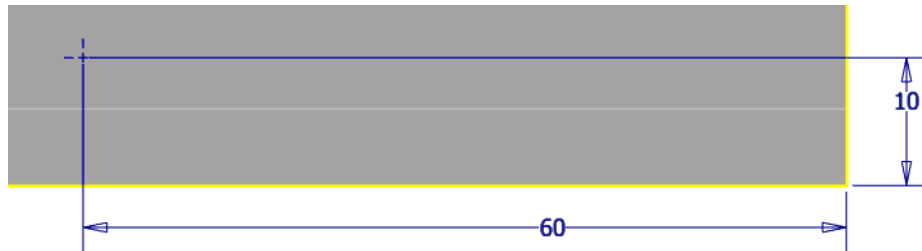
10. For the second dimension, click the point then click the left edge of the block.

11. Drag the dimension down then click to place the dimension.

12. Enter **40** then click OK.



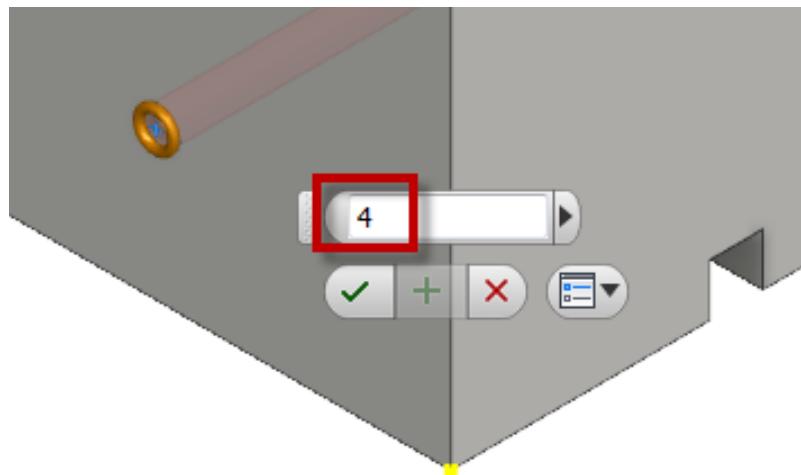
13. Repeat this workflow to add the dimensions for the point on the right of the block. The dimensions for the right side are **10mm** and **60mm**.



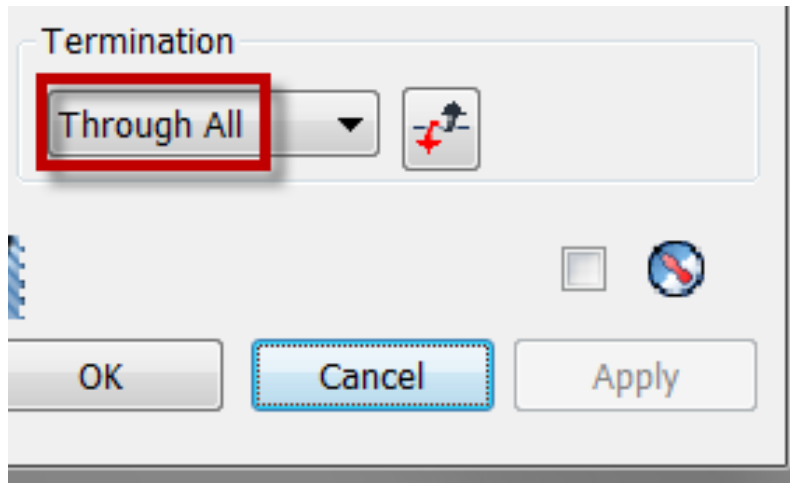
14. On the ViewCube, click Home.

15. Press **H**. This is the shortcut key for the Hole tool.

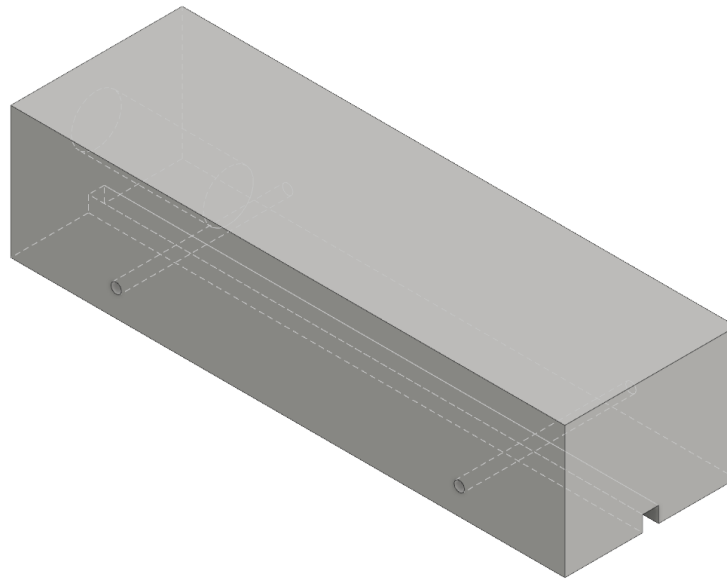
16. For Diameter, enter **4**.



17. Under Termination, the default setting Through All should be displayed.



18. Click OK. The 2 holes are created.



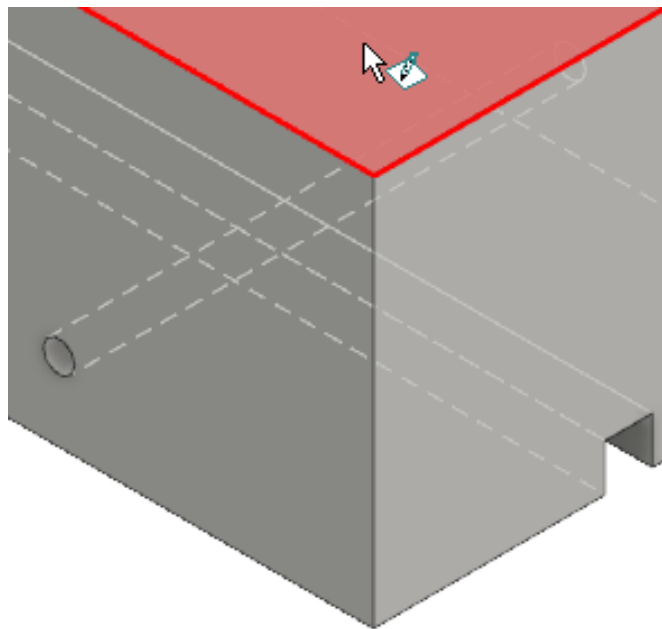
### Step 3: Creating the Wheel Openings

To create space for the wheels you subtract material from the block. To do that you create sketches then cut the material using the extrude tool.

1. On the Sketch panel, click Start 2D Sketch.



2. Select the top face of the block as shown.

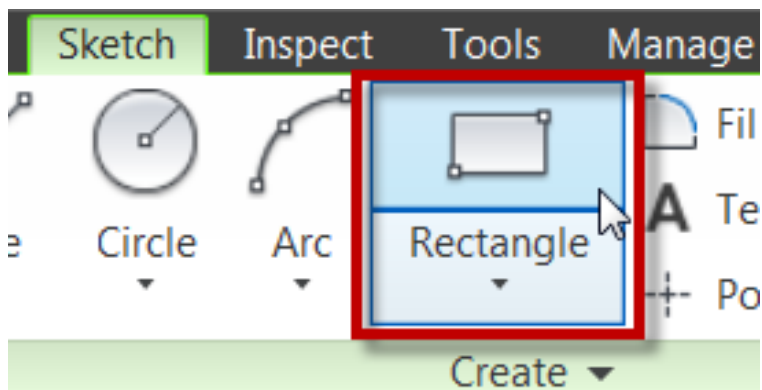


3. On the ViewCube, click Top then click the arrow to rotate the block.

4.

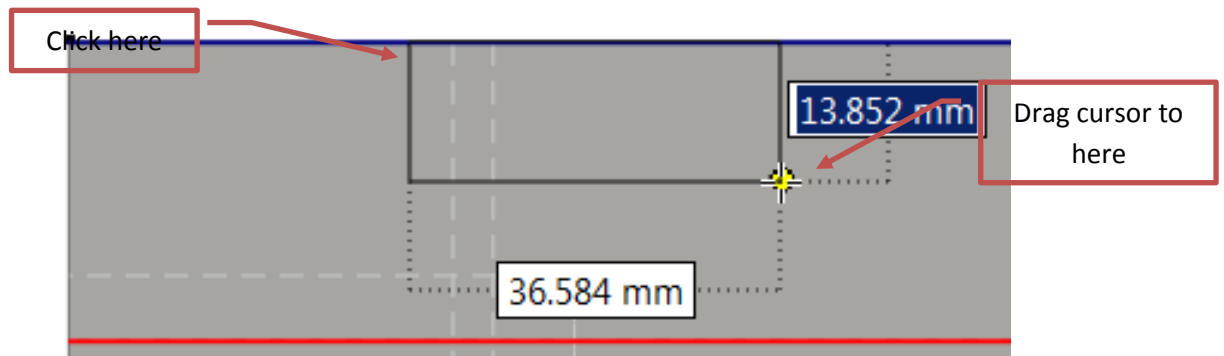


4. On the Create panel, click Rectangle.

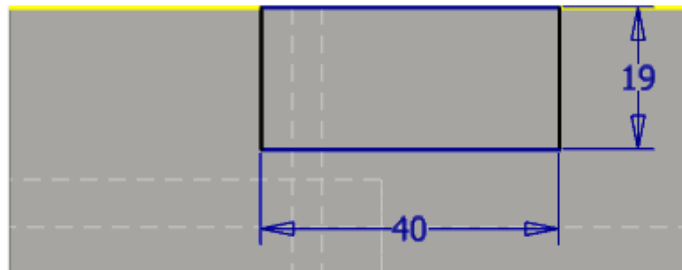




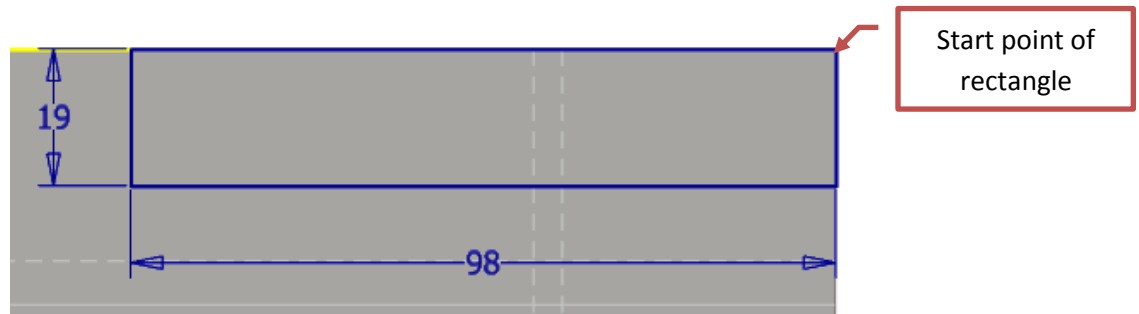
5. Click on the top left edge of the face above the hole then drag down and to the right.



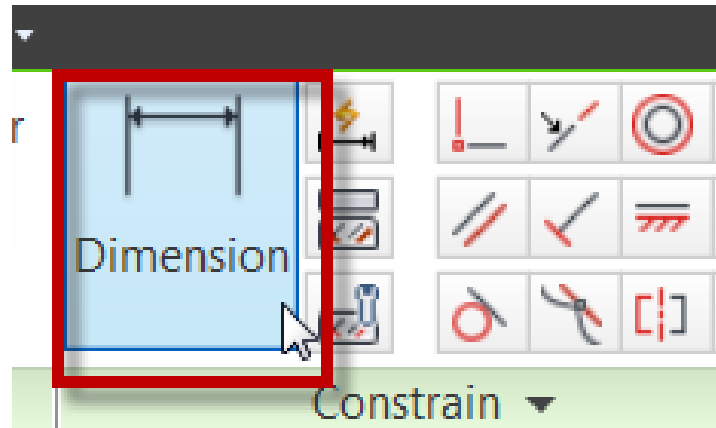
6. Enter **19** then press Tab and enter **40**. Press Enter, right-click, and then click OK.



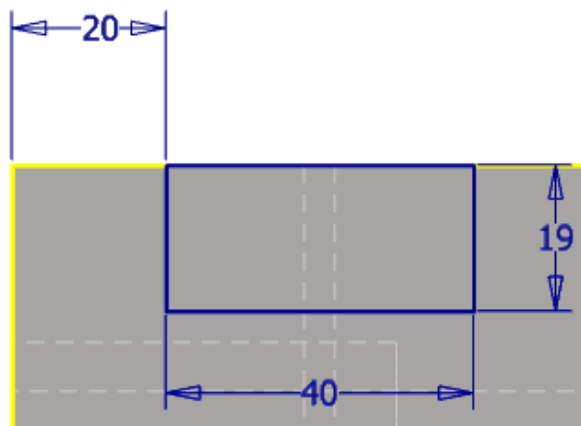
7. Draw a second rectangle **19mm x 98mm** starting at the top right corner of the sketch.



8. On the Constrain panel, click Dimension.

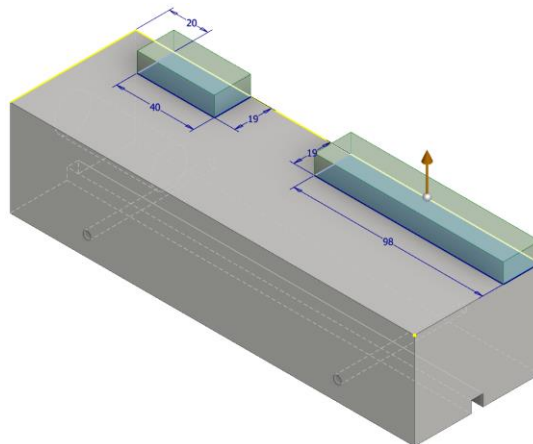


9. Add a **20mm** dimension to the first rectangle as shown.

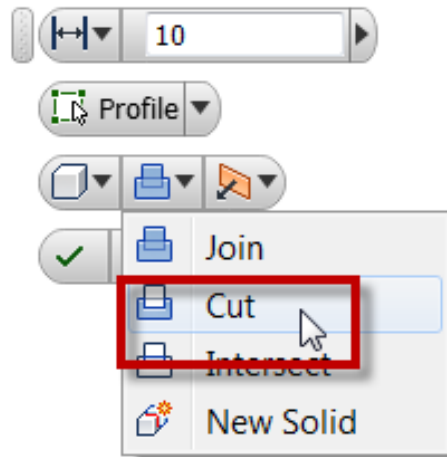


10. Press **E**.

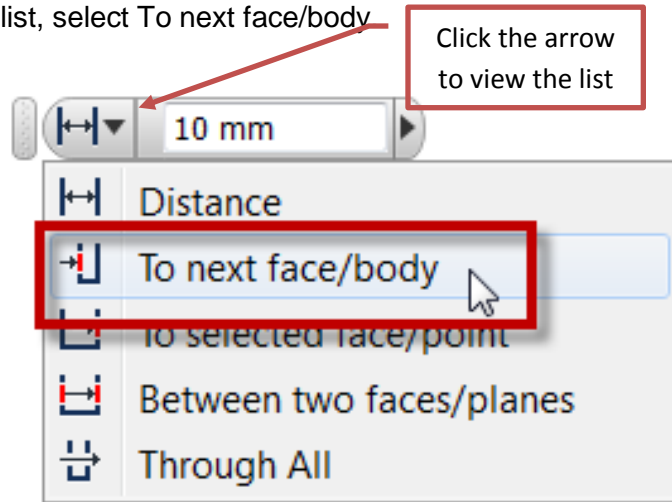
11. Select the 2 profiles as shown.



12. From the Operation list, select Cut.

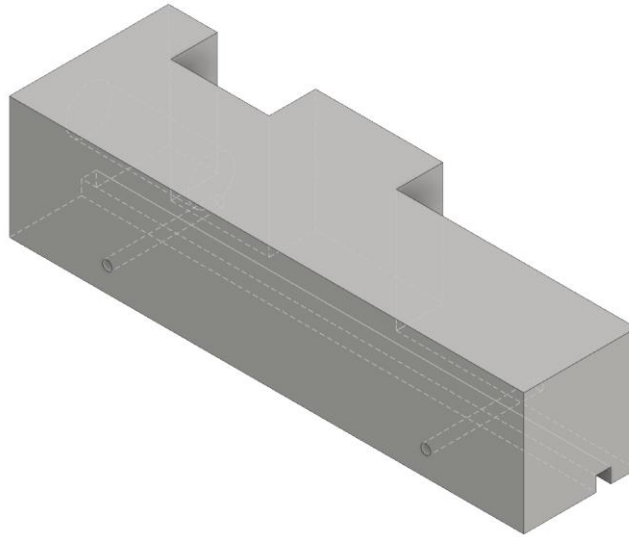


13. On the Extents list, select To next face/body

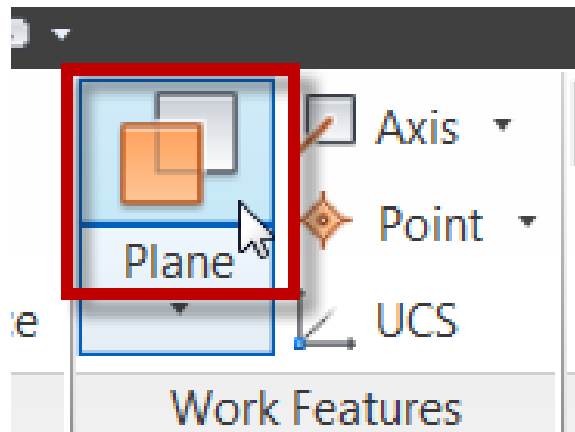


Click the arrow to view the list

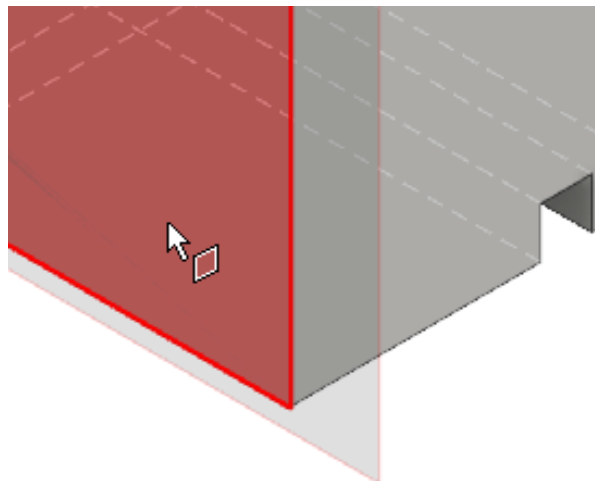
14. Click OK.



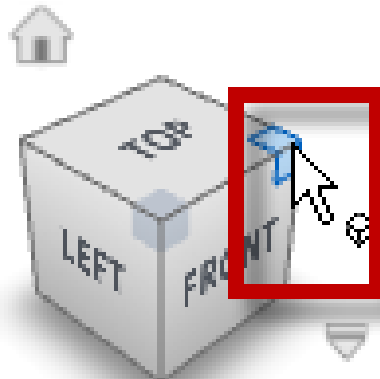
15. On the Work Features panel, click Plane.



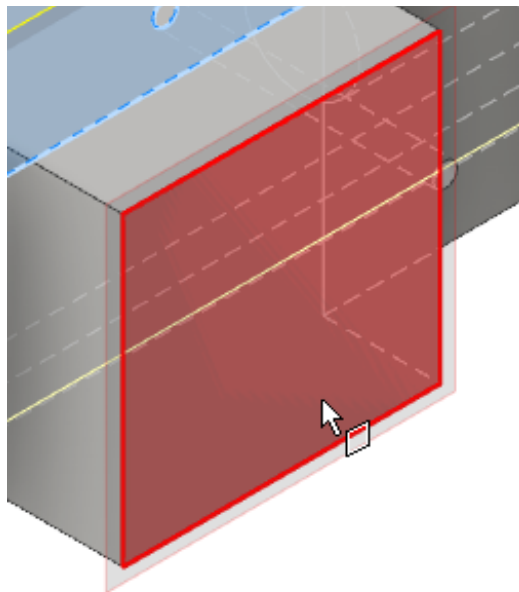
16. Select the front face of the block.



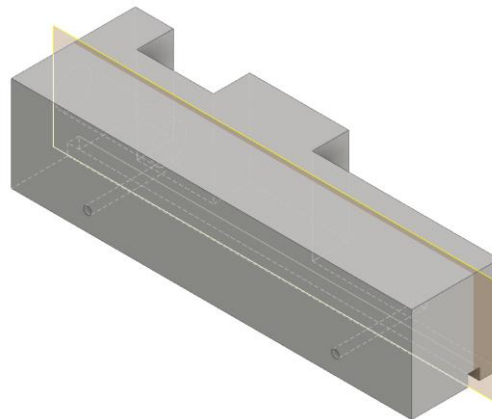
17. On the ViewCube, click the top right corner.



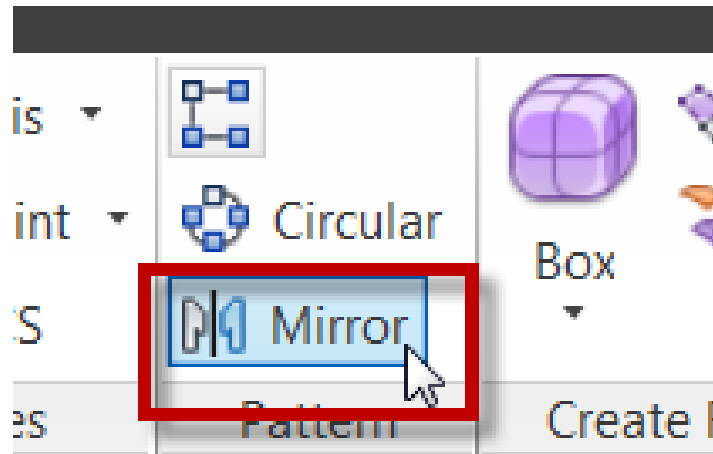
18. Select the face as shown. The work plane is created through the center of the block.



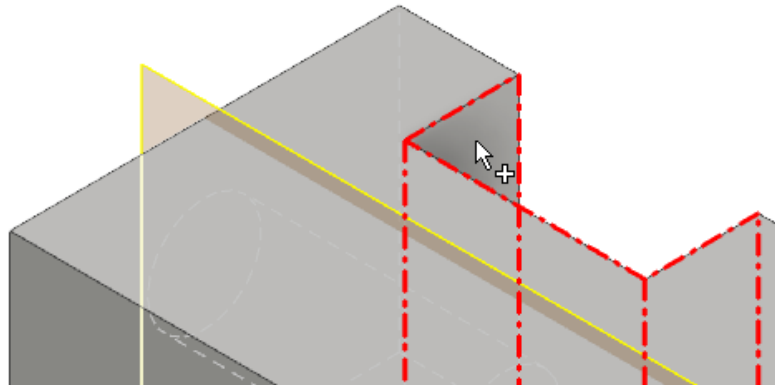
19. On the ViewCube, click Home.



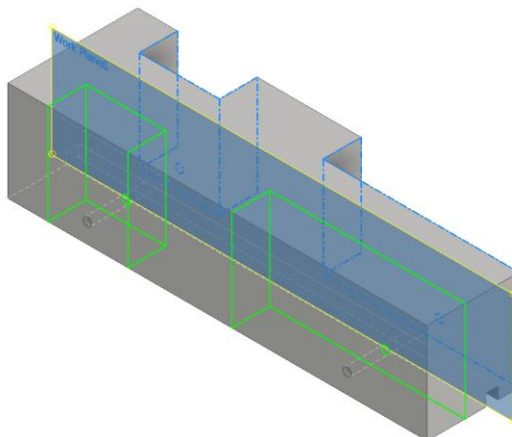
20. On the Pattern panel, click Mirror.



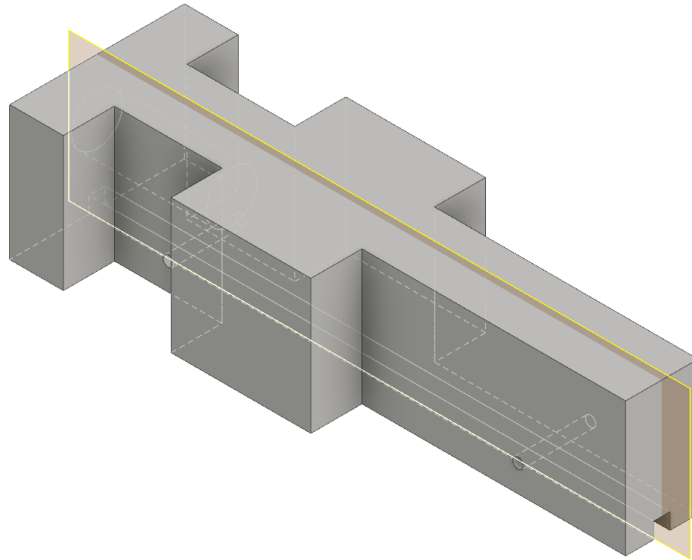
21. Select the extruded feature as shown.



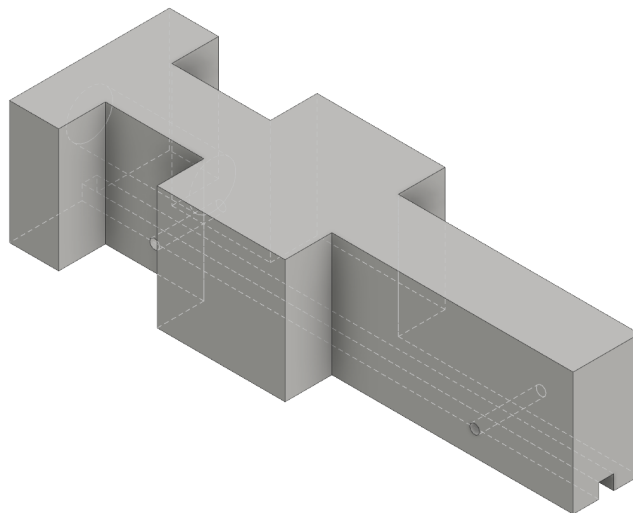
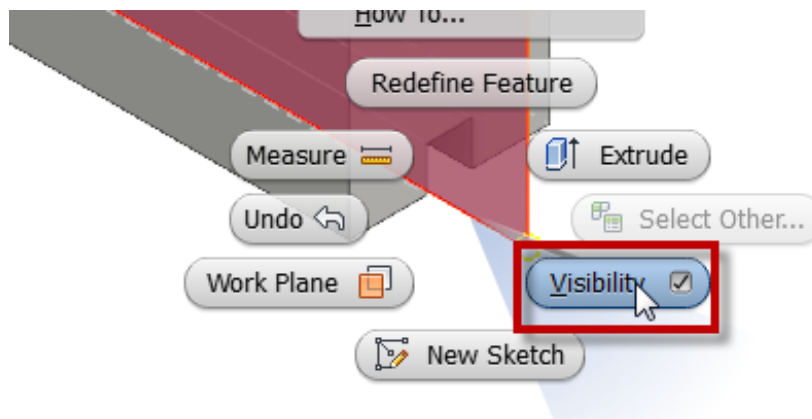
22. In the Mirror dialog box, click Mirror Plane then select the work plane.



23. Click OK.



24. In the graphics window, right-click the work plane, then click Visibility.



25. Save the file.

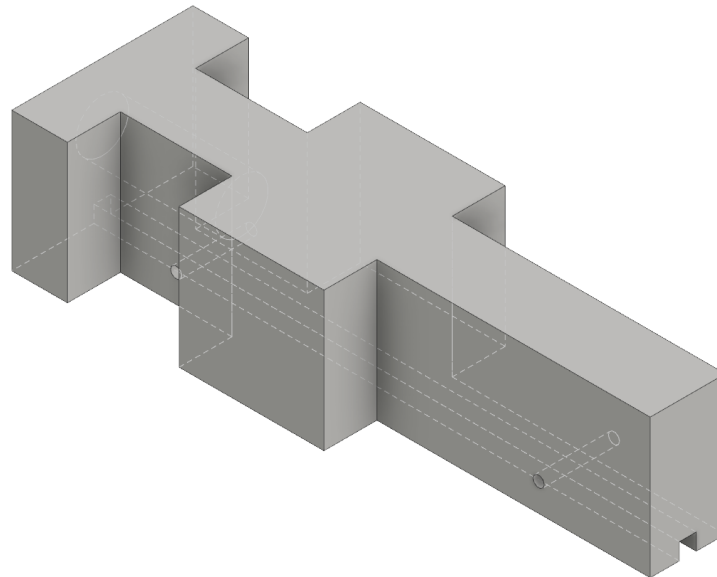
### Activity 3: Model the F1 in Schools Car Body Shape

In this section of the tutorial, you will modify the body, and then model the front and rear wing parts for your F1 car design. Throughout the process, refer to the technical regulations to check Regulation compliance. For video instructional support, download the step-by-step video tutorials for this activity.

#### Step 1: Create Freeform Plane

To modify the block, create a freeform plane on the top face of the block. The plane can be modified using the edit form tool to quickly create shapes that are more visually compelling.

1. The F1 Model Block from the previous activity should be open.

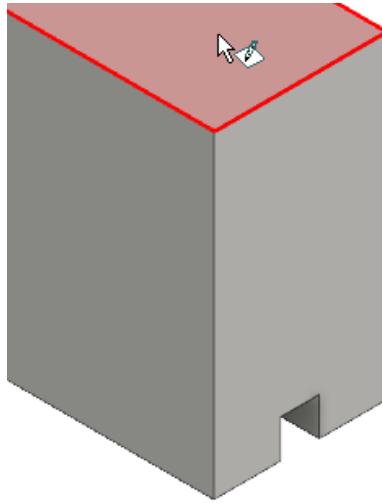


2. On the Sketch panel, click Start 2D Sketch.

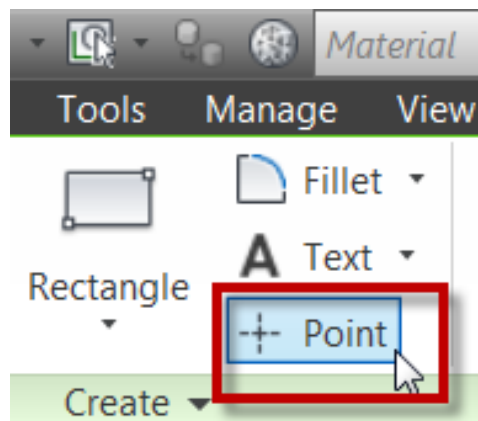




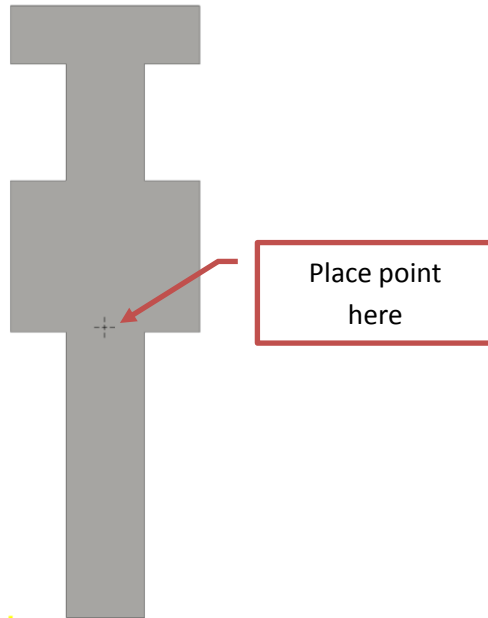
3. Click the top face as shown.



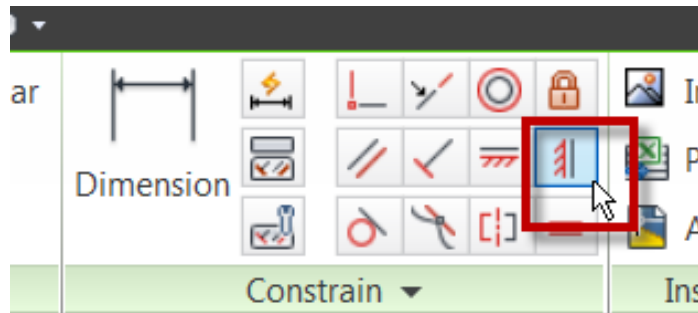
4. On the ViewCube, click Top.
5. On the Create panel, click Point.



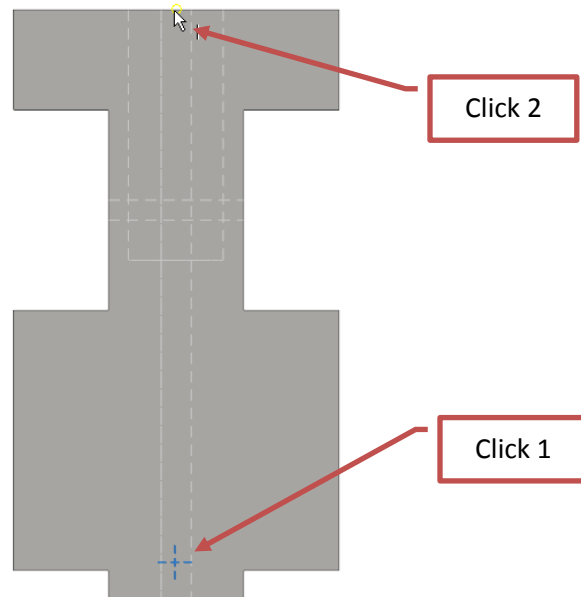
6. Click on the top face near the center of the block.



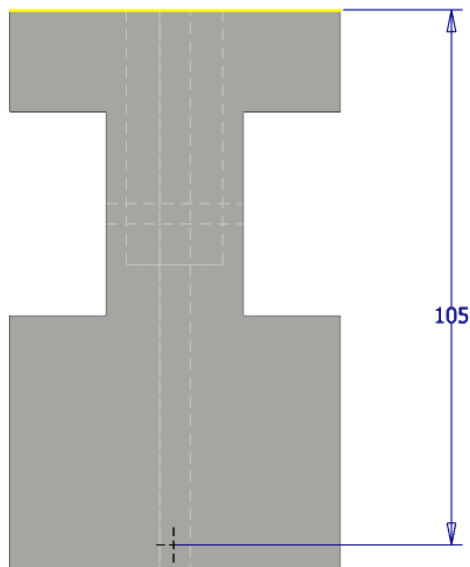
7. On the Constrain panel, click Vertical.



- Click the point (1) then click the midpoint (2) on the top edge of the block. Make sure you select the midpoint, not the endpoints of the tether slot.

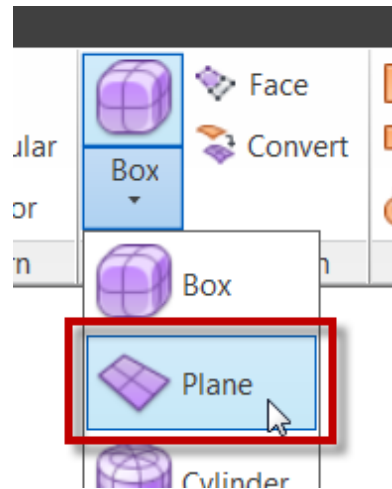


- On the Constrain panel, click Dimension.
- Add a **105** dimension between the point and the top edge of the block.



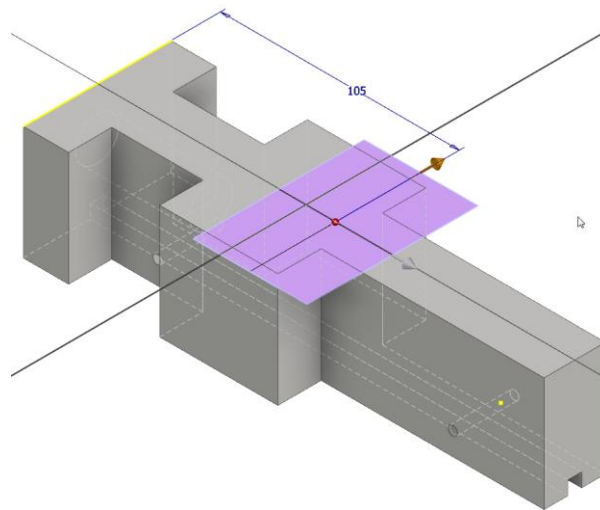
- On the Exit menu, click Finish Sketch.
- On the ViewCube, click Home.

13. On the Freeform panel, from the list, click Plane.



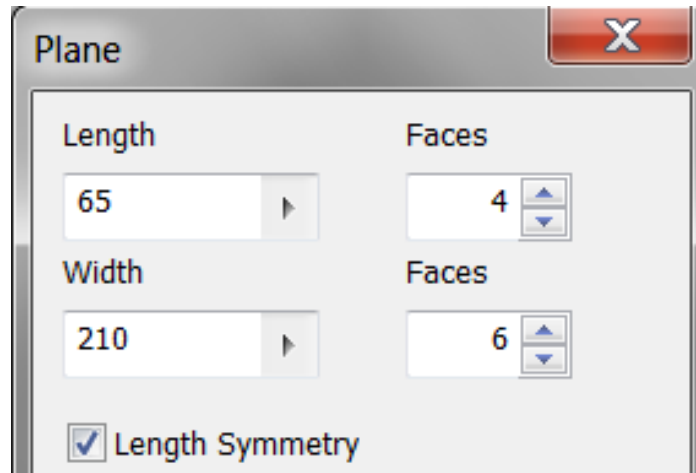
14. Click the top surface of the block.

15. For the center of the plane, click the point created in the previous steps.



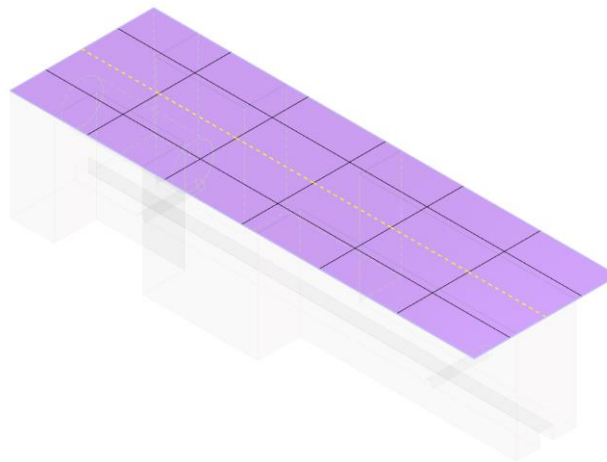
16. In the Plane dialog box, enter the following values;  
a. For Length, enter **65** then for Faces enter **4**.  
b. For Width, enter **210** then for Faces enter **6**.

c. Select Length Symmetry.



17. Click OK. 6 x 4 faces give us enough control over the geometry without being too dense.

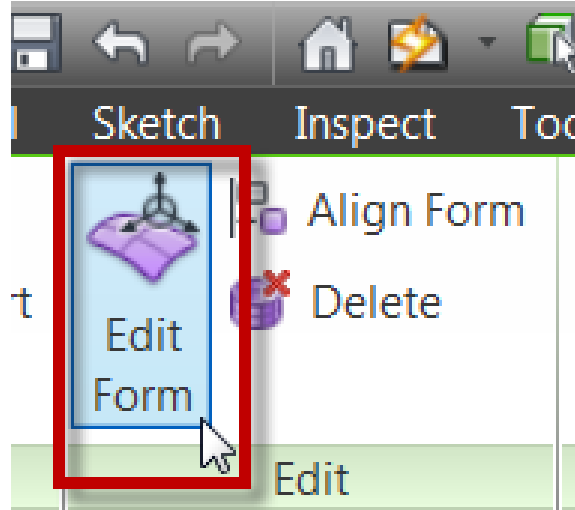
18. In the browser, right-click the sketch above the Form entry, then click Visibility.



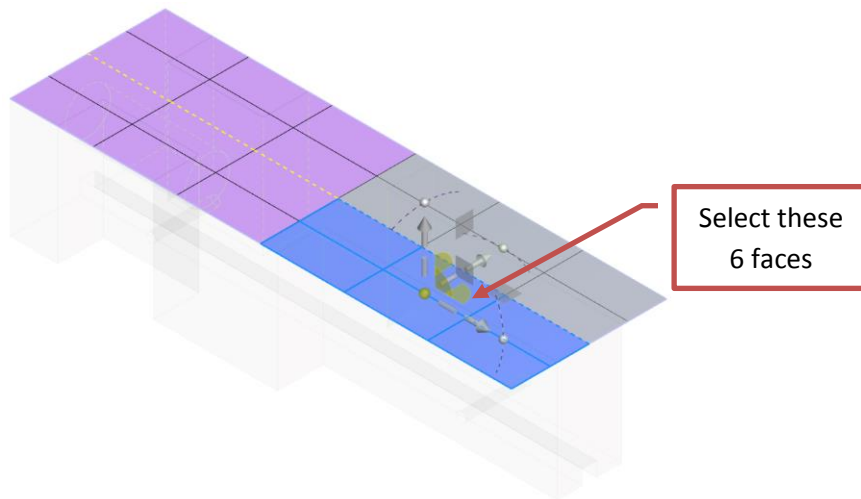
## Step 2: Edit the Freeform Plane

The edit form tool allows you to directly manipulate and modify a freeform shape. In this example, you select faces then drag the manipulators to create an aerodynamic shape for your F1 car.

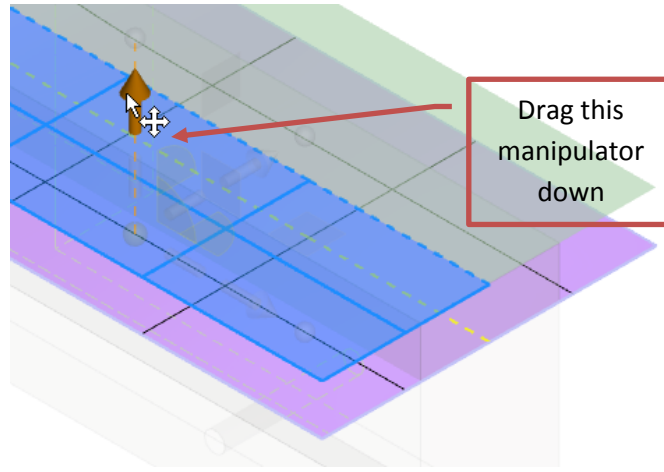
1. On the Edit panel, click Edit Form.



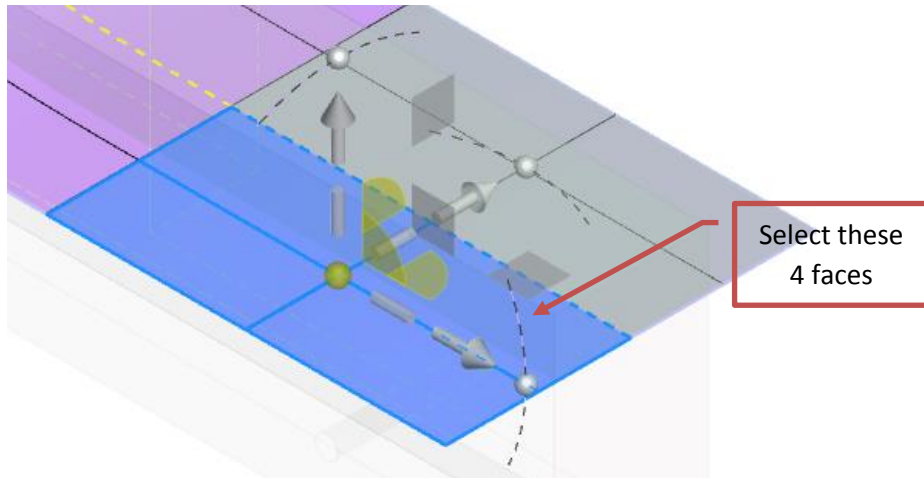
2. Click the 6 faces as shown. To make multiple selections, press Shift while selecting. The 6 faces on the other side of the symmetry line are also selected.



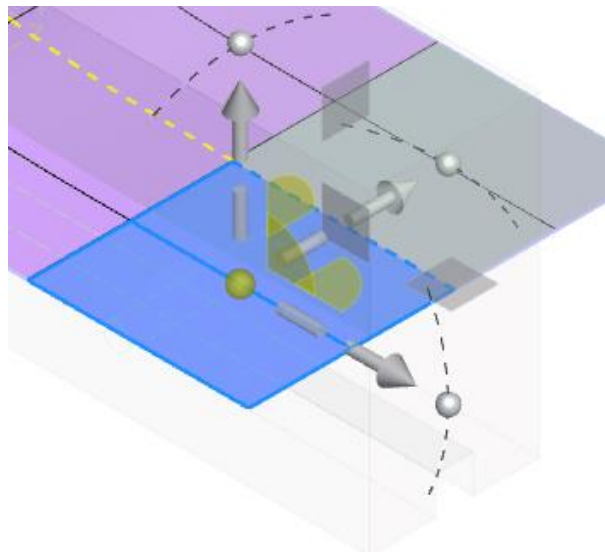
3. Drag the manipulator down then enter **-10**. Do not click OK.



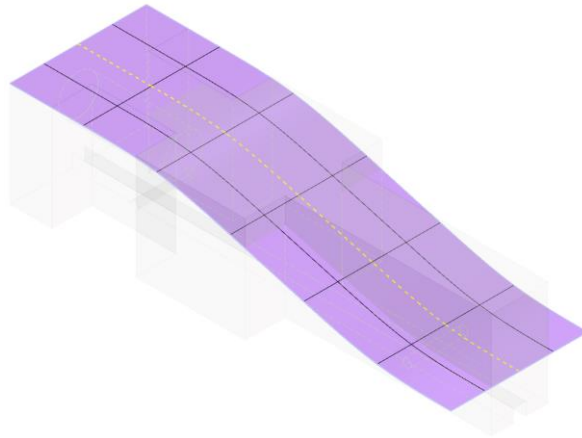
4. Click the 4 faces as shown then drag the manipulator down **-10**.



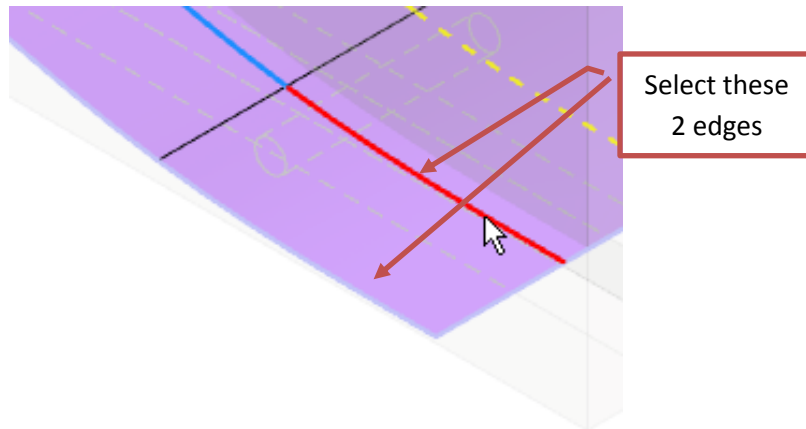
5. Click the 2 faces as shown then drag the manipulator down **-10**.



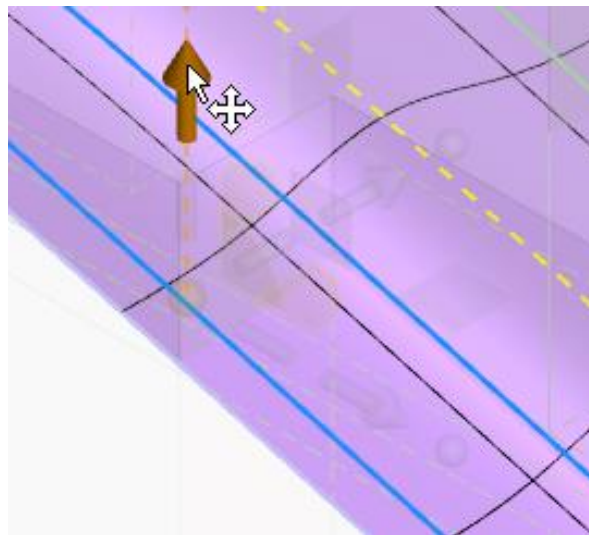
6. Click OK.



7. Double-click the 2 edges as shown. The complete edges should be selected. To make multiple selections, press Shift while selecting.

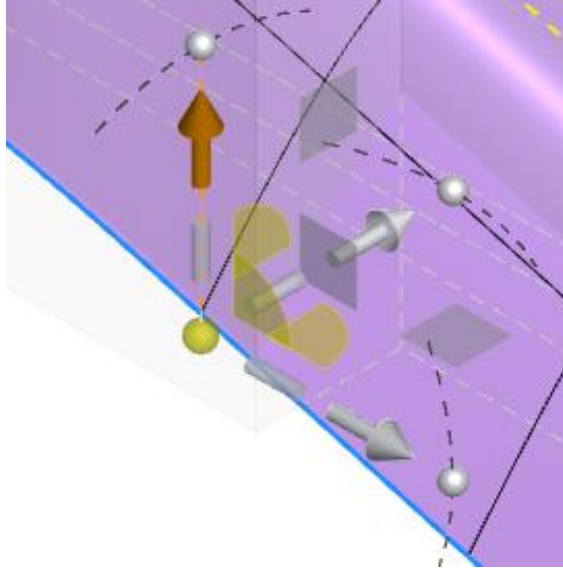


8. Click Edit Form then drag the manipulator down **-10**.

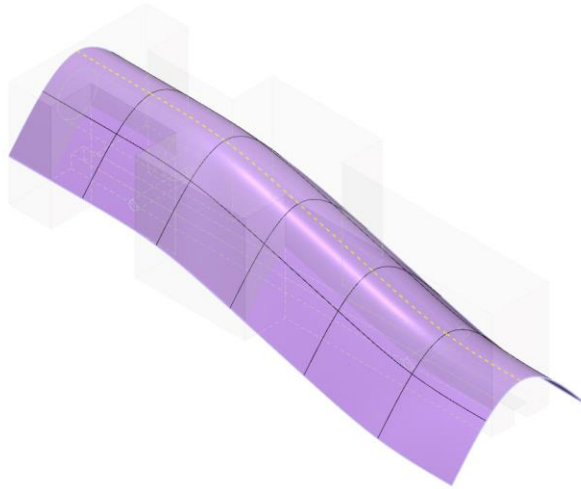




9. Select the outside edge only then drag the manipulator down **-20**.

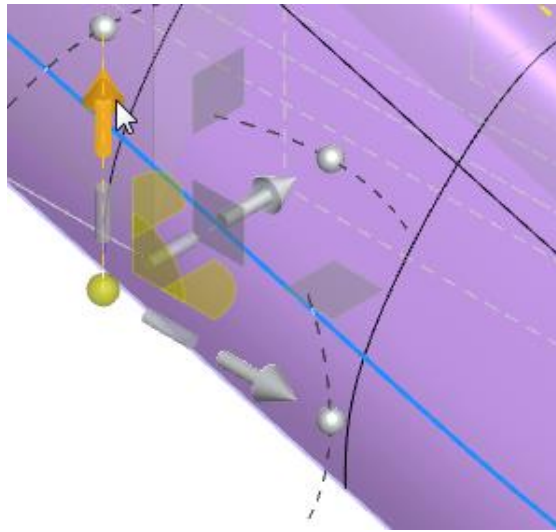


10. Click OK.

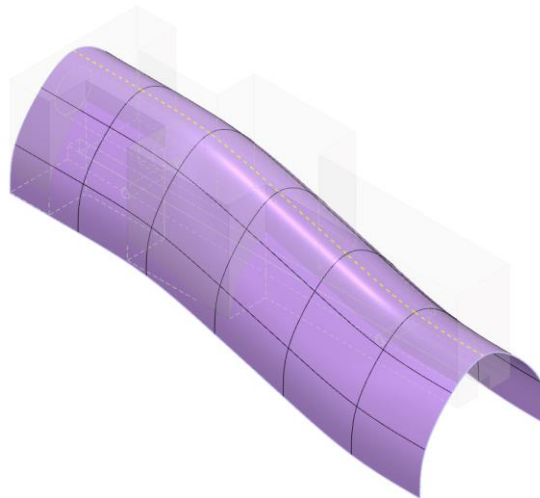


11. On the Edit panel, click Edit Form.

12. Double-click the outside edge, then hold down the Alt key and drag the manipulator down **-20**. Holding down Alt adds new faces to the form.

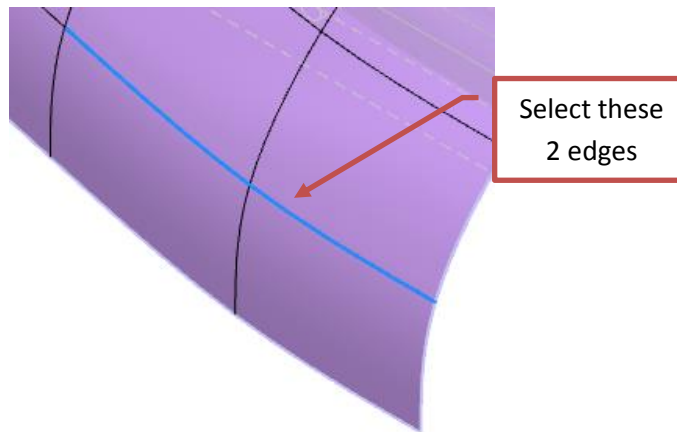


13. Click OK.

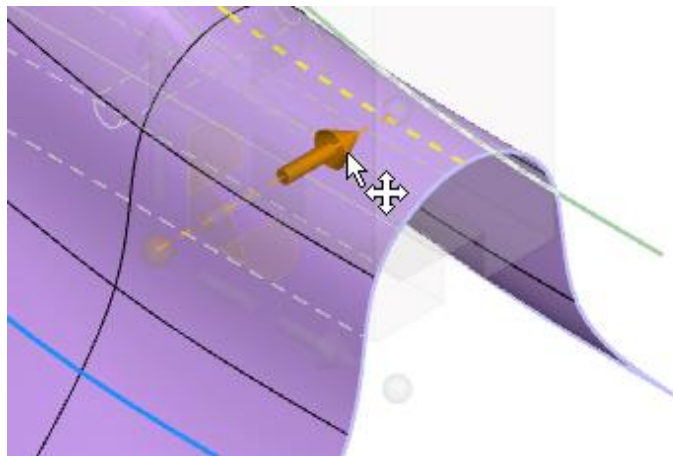


14. On the Edit panel, click Edit Form.

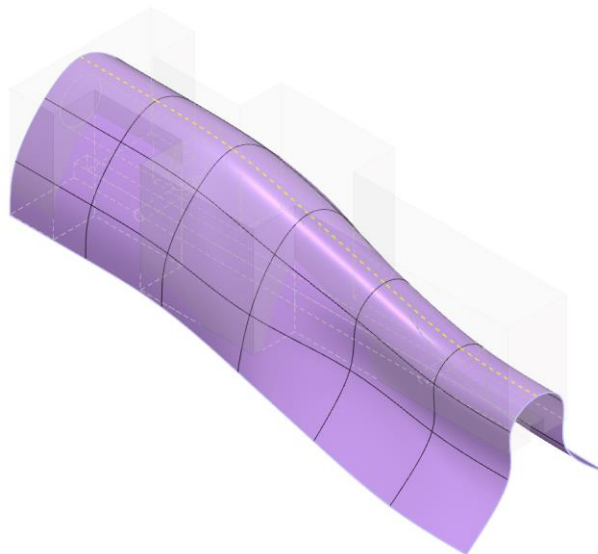
15. Select the 2 edges at the front of the plane as shown.



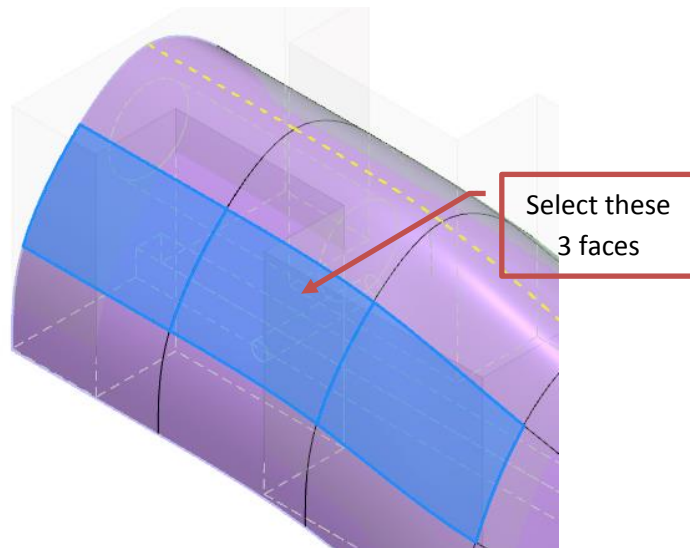
16. Drag the manipulator into the car **25**.



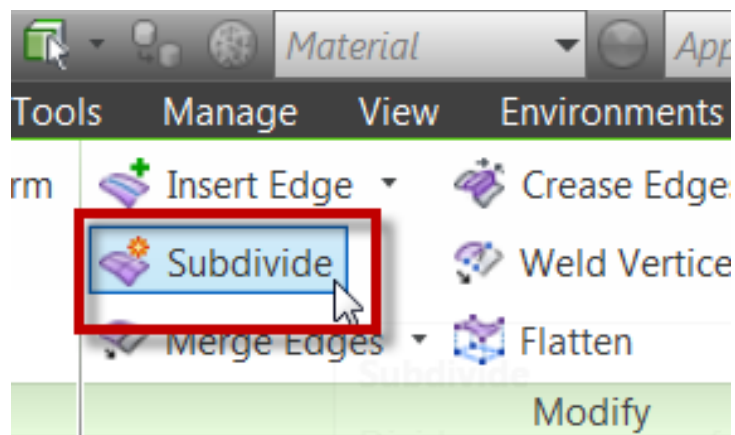
17. Click OK.



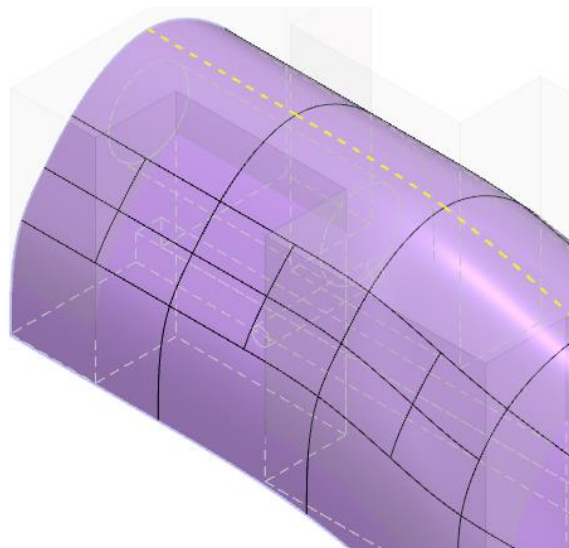
18. Select the 3 faces as shown.



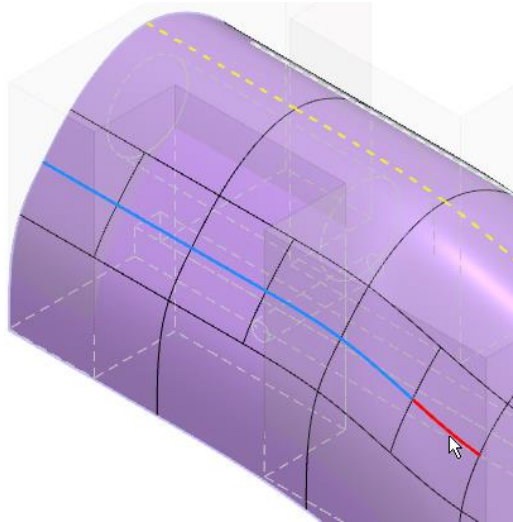
19. On the Modify panel, click Subdivide.



20. Click OK.

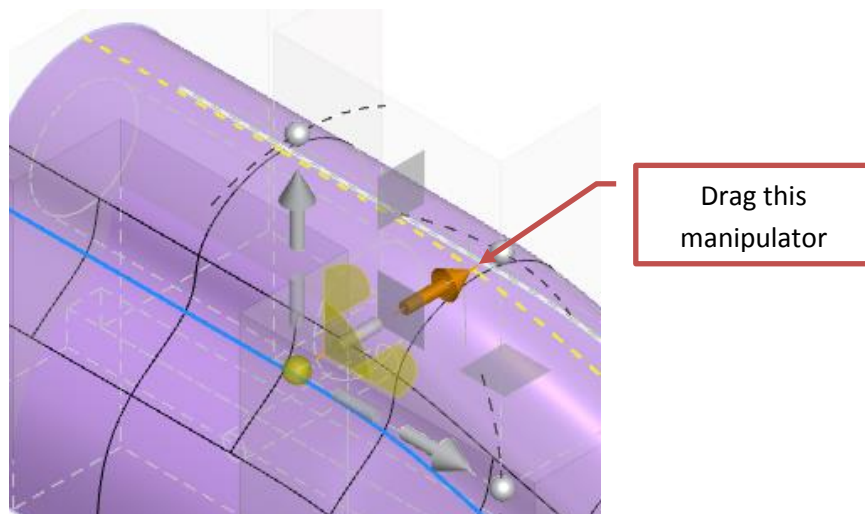


21. Double-click the edge as shown. The complete edge should be selected.

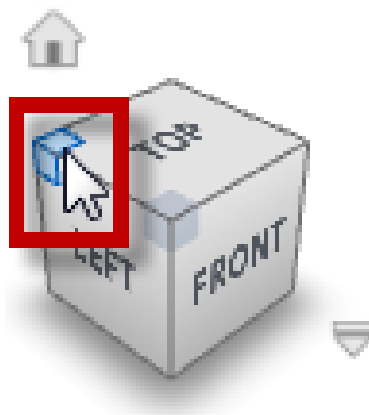


22. On the Edit panel, click Edit Form.

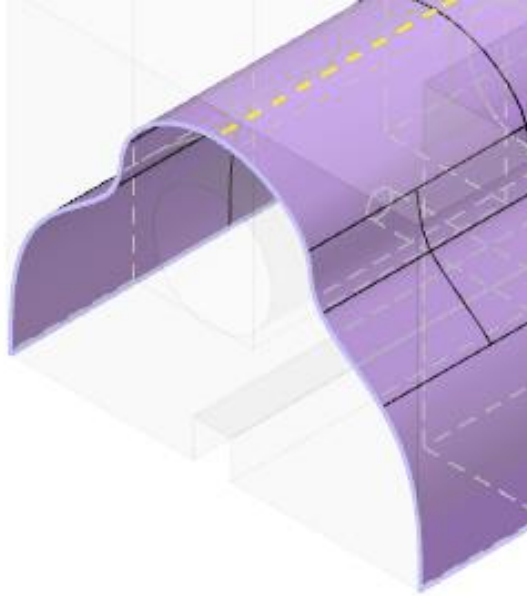
23. Drag the manipulator into the car **10mm**.



24. On the ViewCube, click the top left corner.

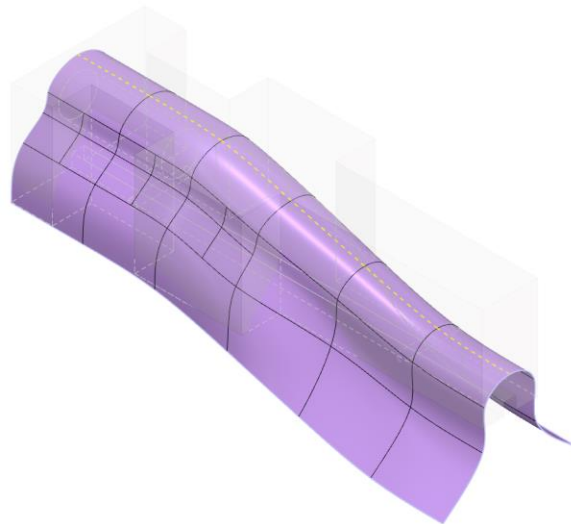


25. Review the distance between the cartridge chamber and the modification you just created. There must be 3.5mm of material between them per the Regulations



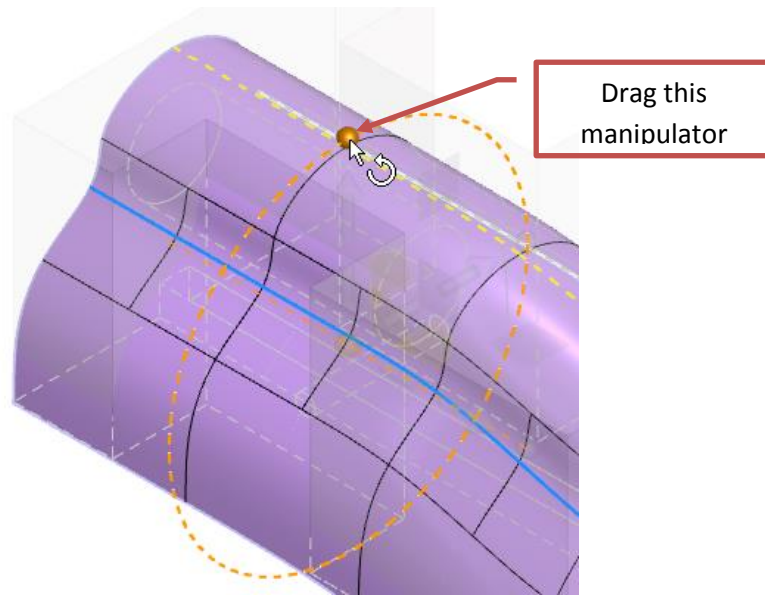
26. Click OK.

27. Return to the Home view.

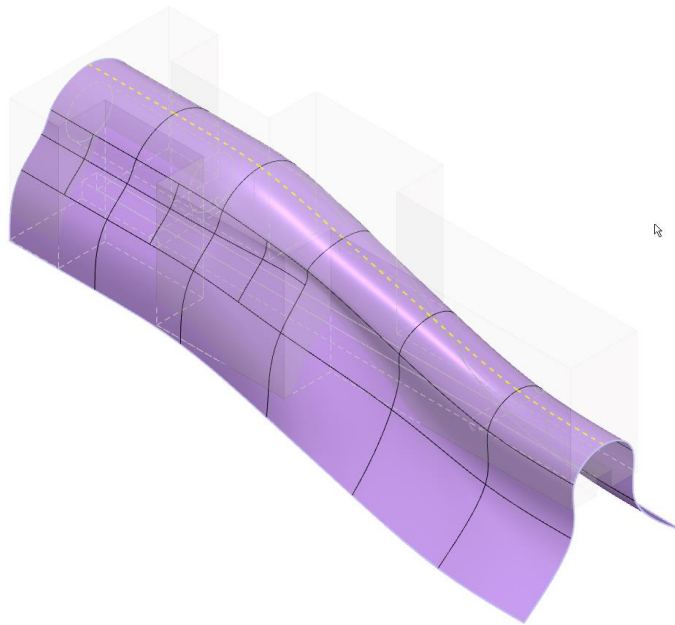


28. On the Edit panel, click Edit Form.

29. Double-click the same edge and drag the rotation manipulator counter-clockwise **70** degrees.



30. Click OK.



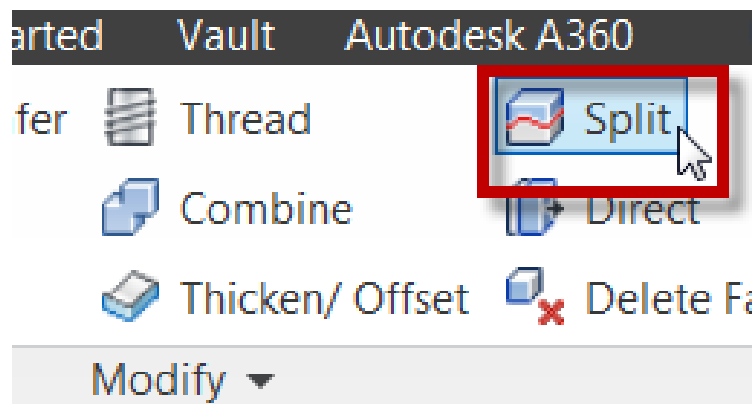
31. On the Exit panel, click Finish Freeform.



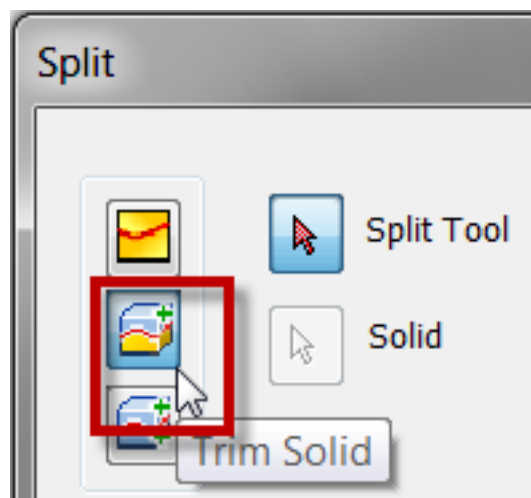
### Step 3: Split the Car Body

The freeform plane you created will be used to remove material above the plane and create a dynamic design for your car. To do this, use the split tool.

1. On the Modify menu, click Split.

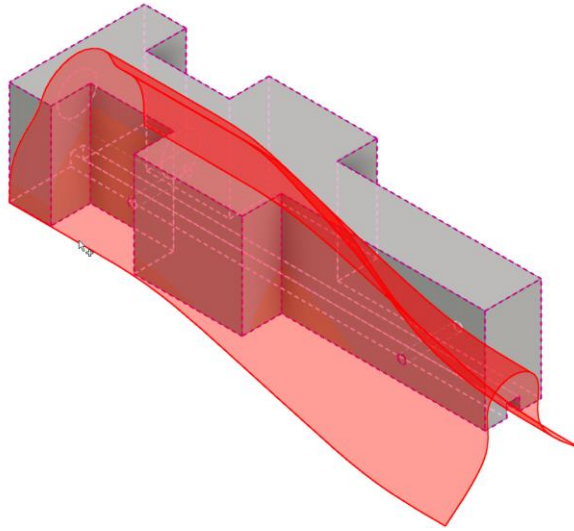


2. On the Split dialog box, click Trim Solid.

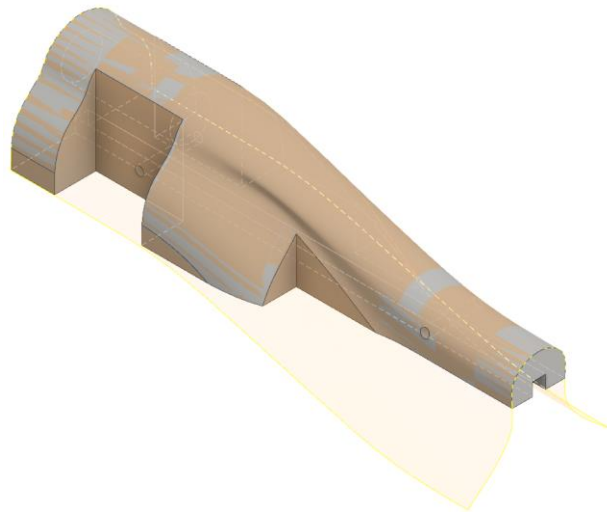




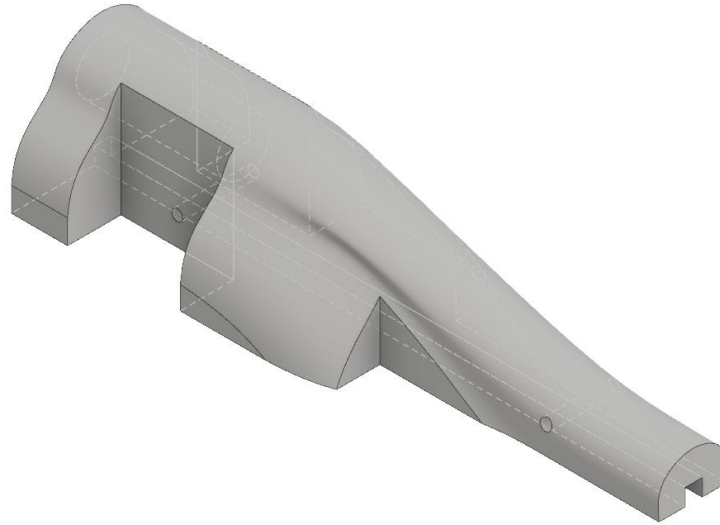
3. Select the freeform surface. An arrow is displayed indicating which side material will be removed from.



4. Click OK.



5. In the browser, right-click the Form entry, then click Surface Visibility.



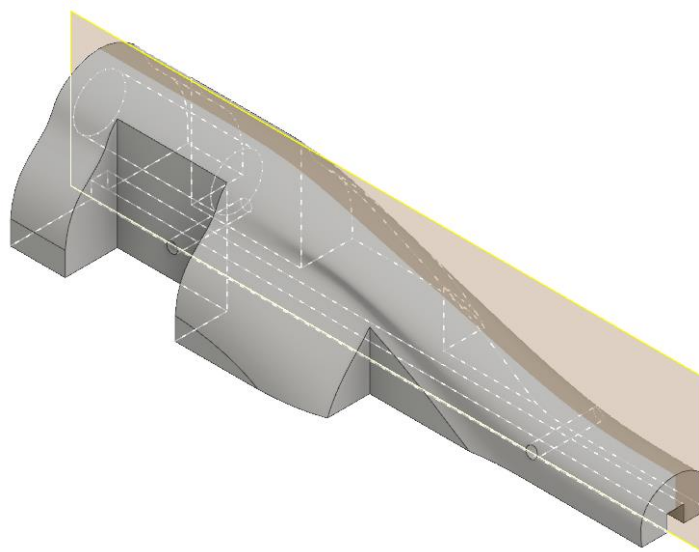
6. Save the file.

### Activity 4: Designing the Wings

In this activity, you will model organic shaped front and rear wings for your car body. The wings are created as solid bodies, which will allow you to 3D print them later. For video instructional support, download the step-by-step video tutorials for this activity.

#### Step 1: Model the Rear Wing

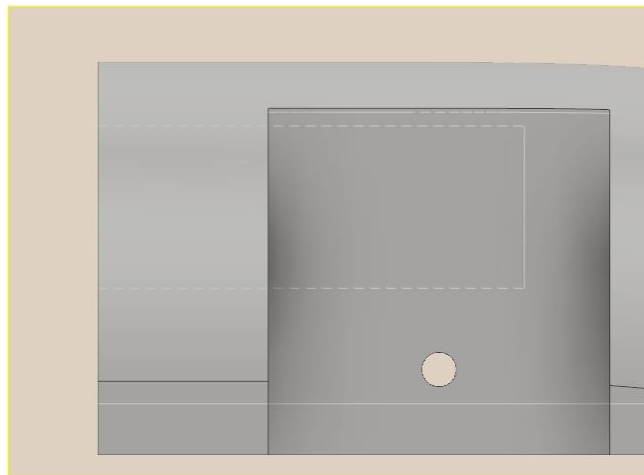
1. In the browser, locate the work plane created previously. Right-click the work plane then click Visibility.



2. On the Sketch panel, click Start 2D Sketch.

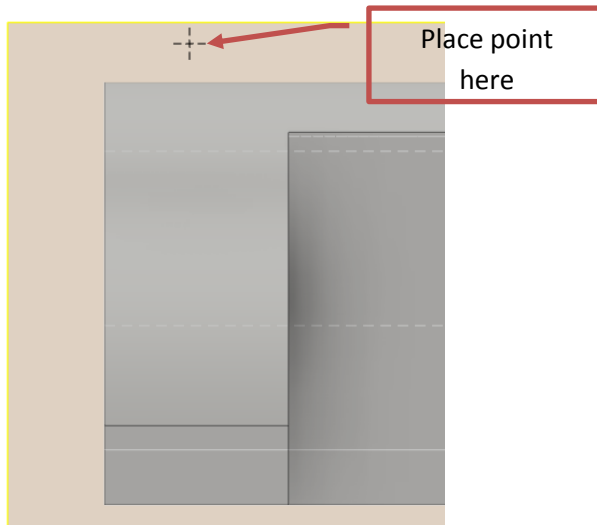


3. Select the work plane.
4. Zoom and pan to the left side of the car.



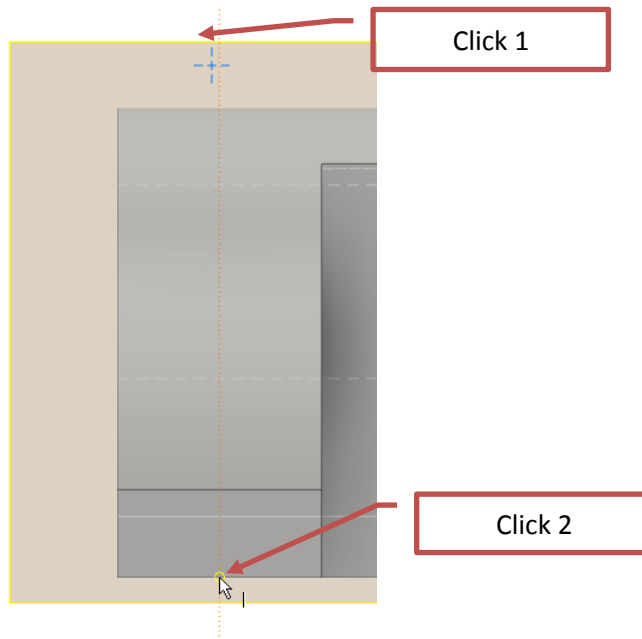
5. On the Create panel, click Point.

6. Click to place a point as shown.

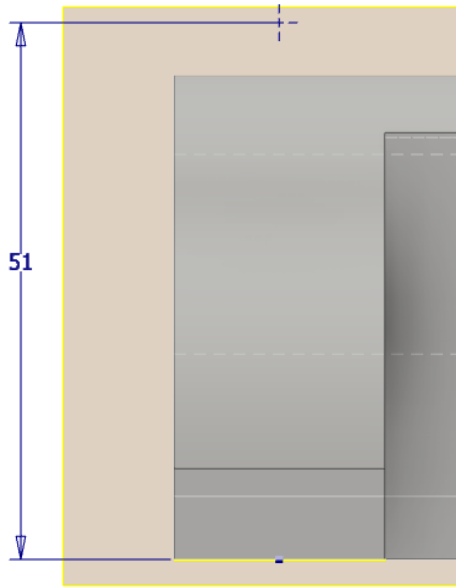


7. On the Constrain panel, click Vertical.

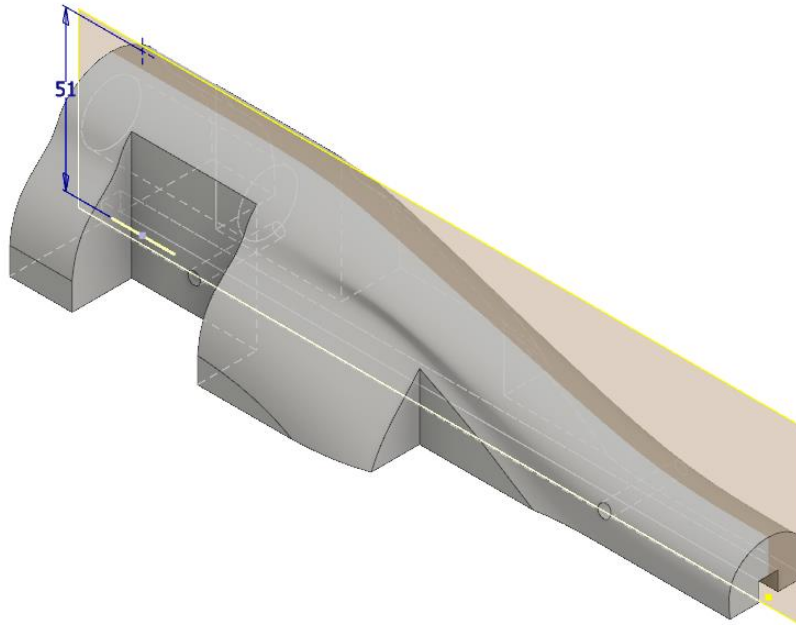
8. Click the point (1) then the center point of the bottom edge (2) as shown.



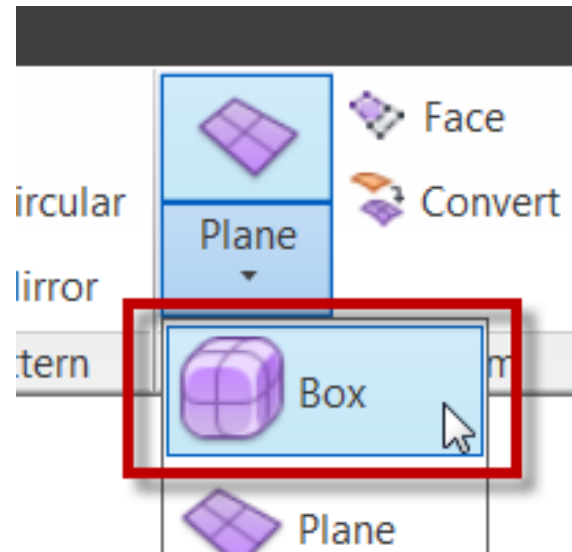
9. Add a dimension of **51mm** between the point and the bottom edge of the block.



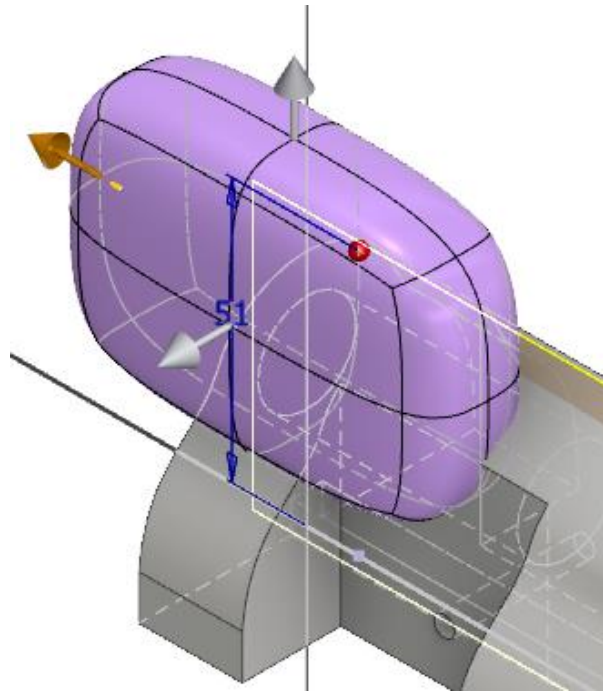
10. On the Exit panel, click Finish Sketch.



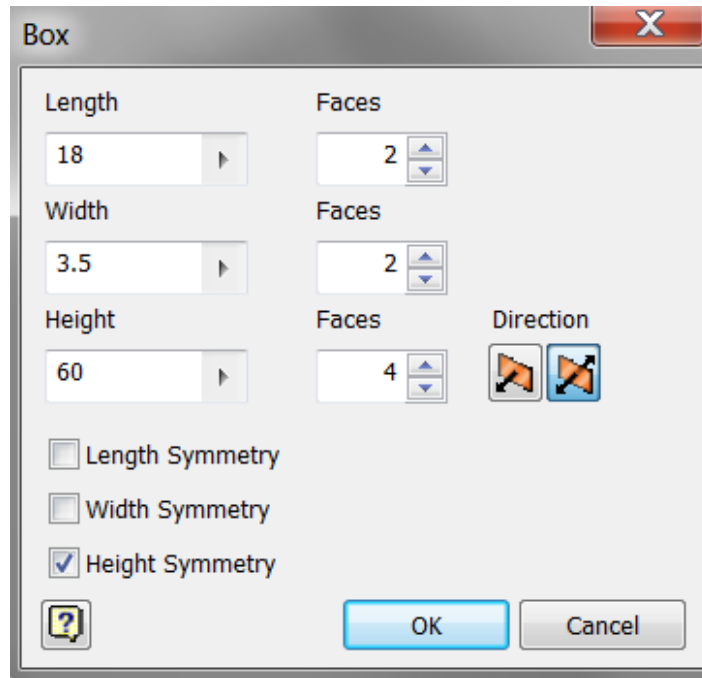
11. On the Create Form panel, click Box.



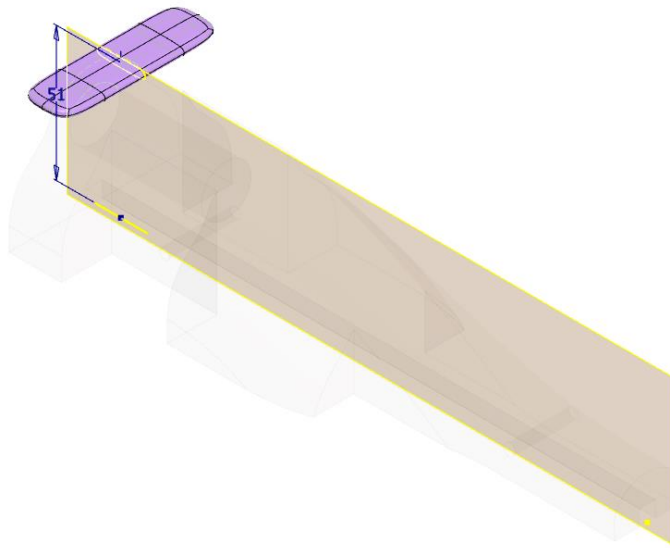
12. Click the work plane then click the point to specify the center of the box.



13. In the Box dialog box, enter the following values;
- For Length, enter **18**.
  - For Width, enter **3.5**.
  - For Height, enter **60** then for Faces enter **4**.
  - Under Direction, select Both Directions.
  - Select Height Symmetry.

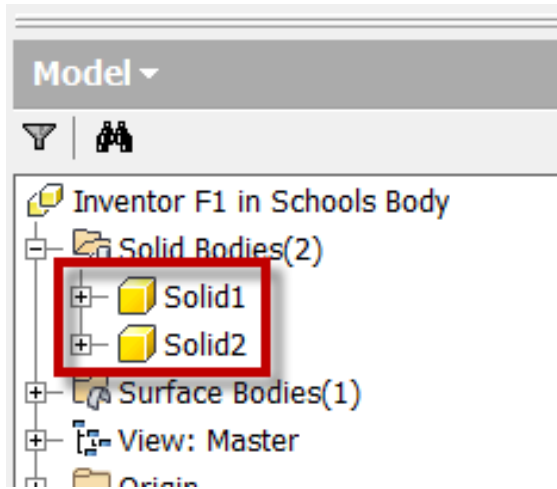


14. Click OK.



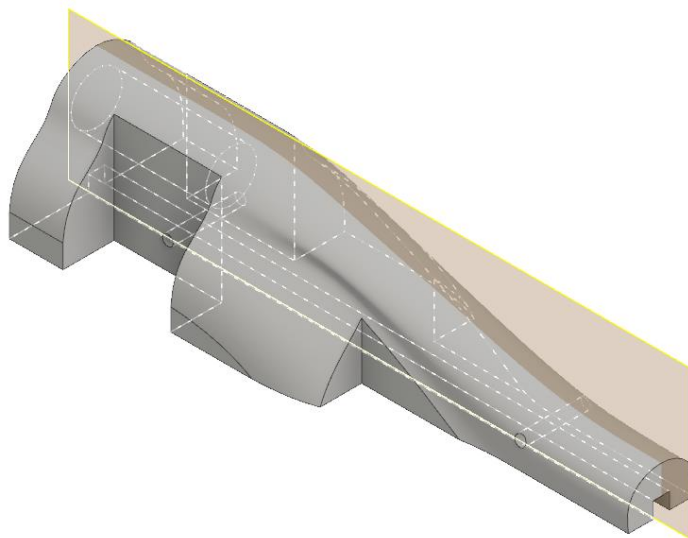
15. On the Exit panel, click Finish Freeform.

16. In the browser, expand Solid Bodies. Note that there are 2 solid bodies, the first is the car body and the second is the rear wing. This will be useful for modeling the parts separately.

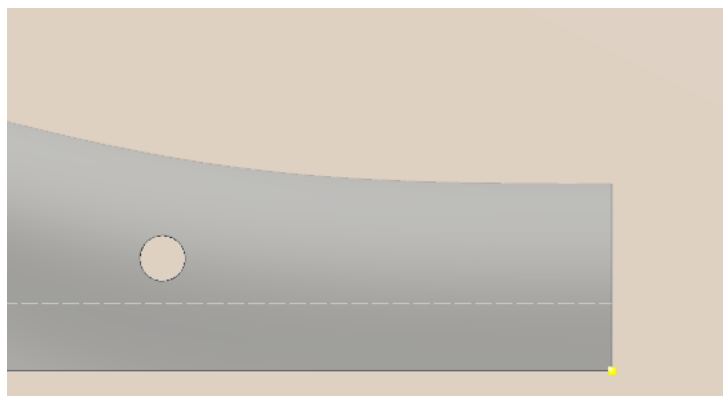


## Step 2: Model the Front Wing

1. In the browser, locate the sketch created previously. Right-click the sketch then click Visibility.

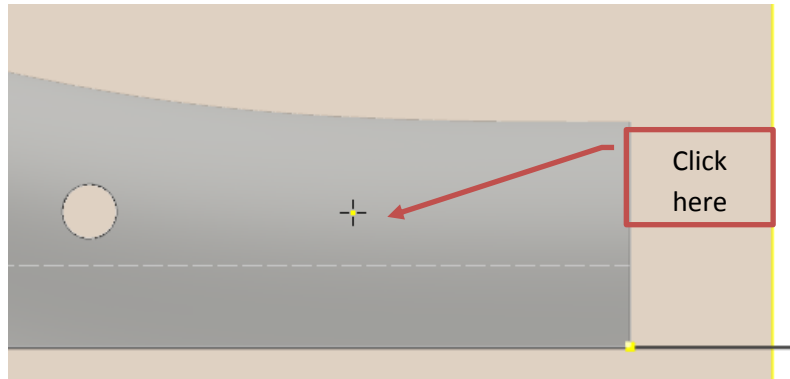


2. On the ViewCube, click Left. Zoom and pan into the front of the car.

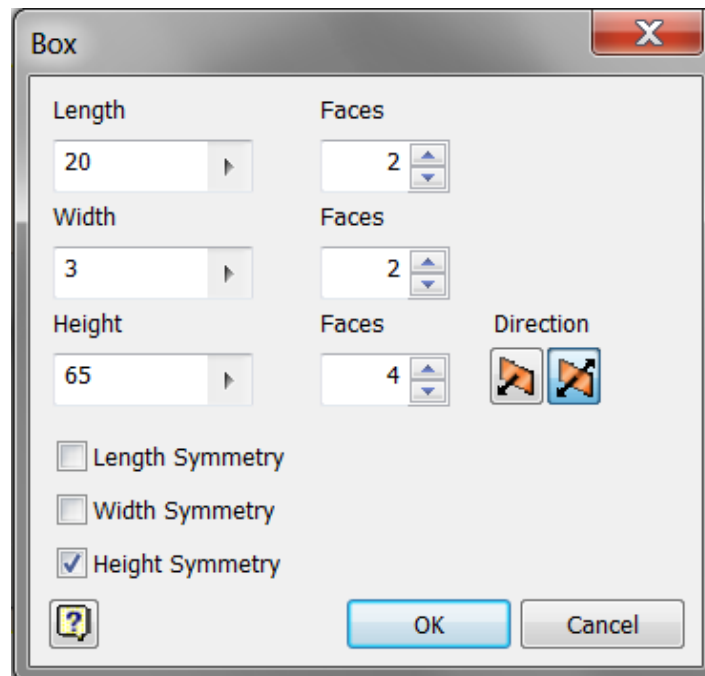




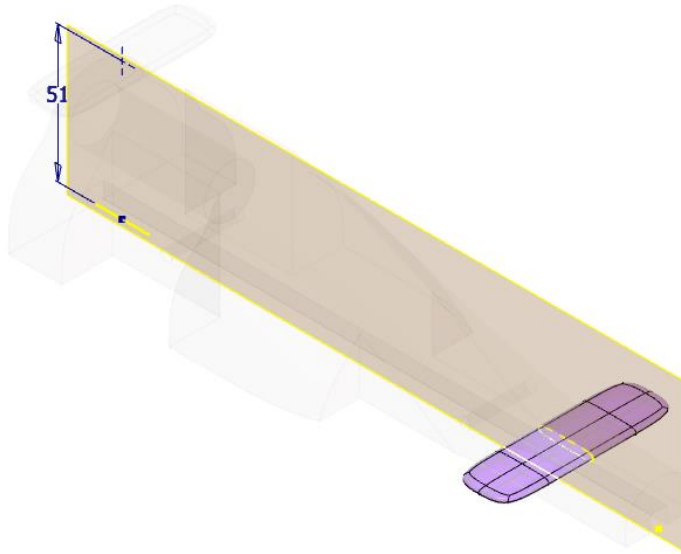
3. On the Create Form panel, click Box.
4. Click the work plane.
5. Click to specify the center of the box. The center is approximately in line with center of the hole and midpoint from the right edge of the block.



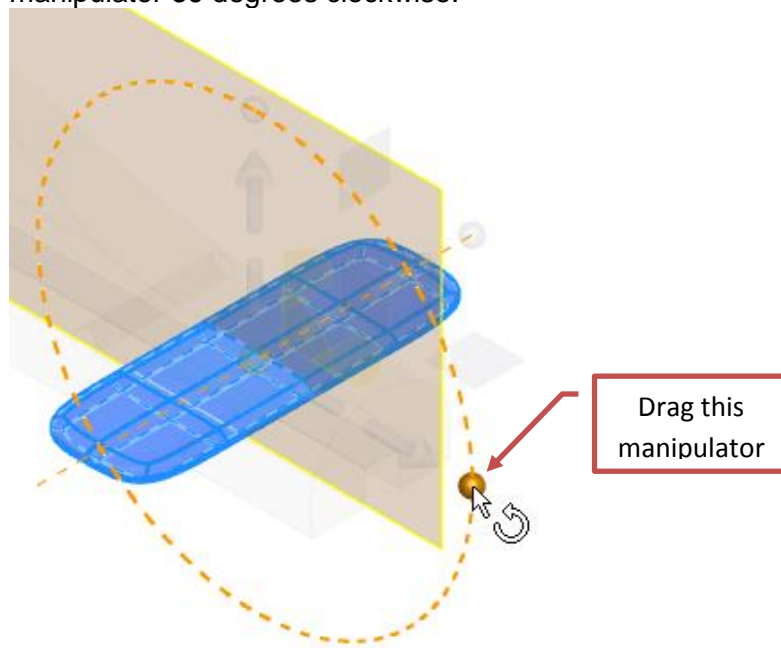
6. In the Box dialog box, enter the following values;
  - a. For Length, enter **20**.
  - b. For Width, enter **3**.
  - c. For Height, enter **65**. For Faces, enter **4**.
  - d. Under Direction, select Both Directions.
  - e. Select Height Symmetry.



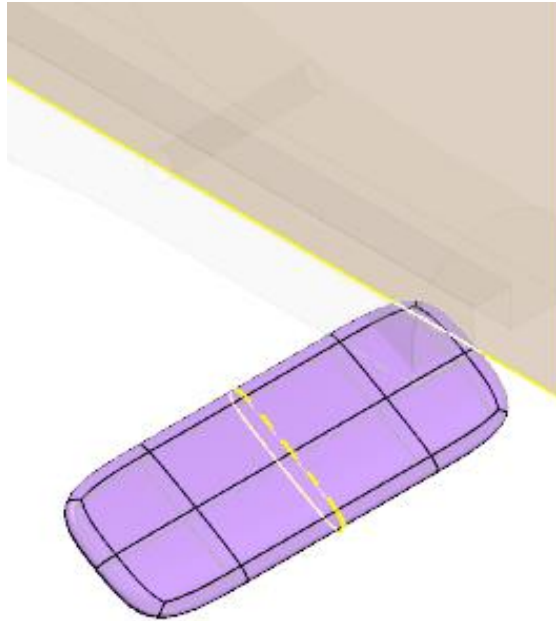
7. Click OK.
8. On the ViewCube, click Home.



9. Double-click a face on the wing. This selects the complete wing.
10. On the Edit panel, click Edit Form.
11. Drag the rotational manipulator **30** degrees clockwise.



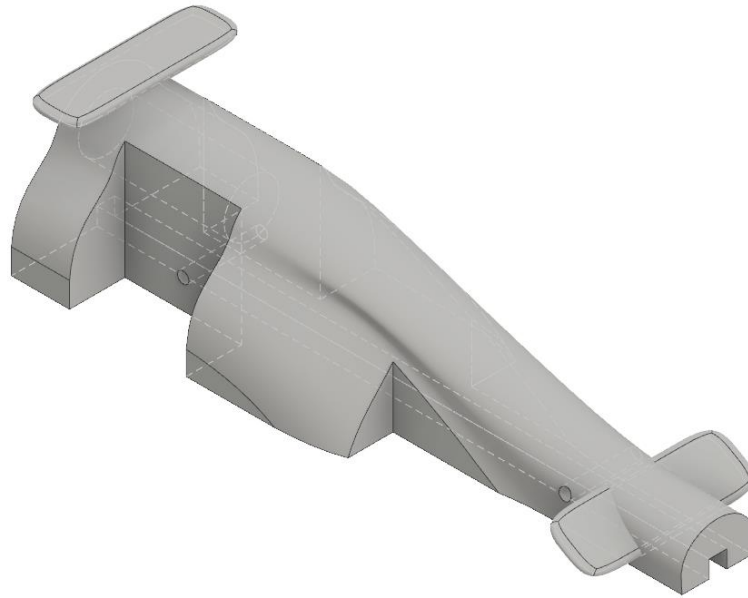
12. Click OK.



13. On the Exit panel, click Finish Freeform.



14. Turn off the visibility of the sketch and the work plane.



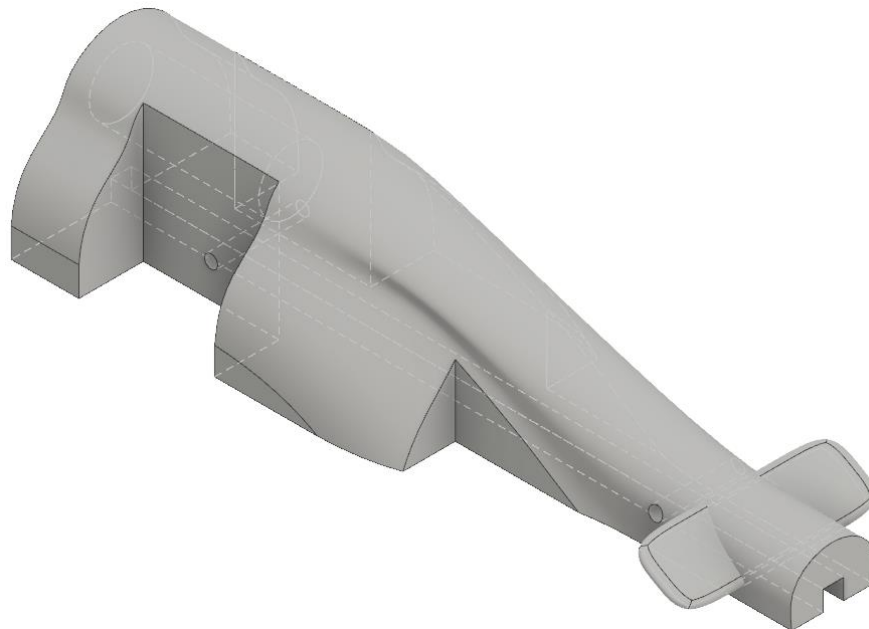
15. Save your file.

### Step 3: Add the Wing Support on the Body

The rear wing is not connected to the car body. You now create a support to connect the car body and the wing.

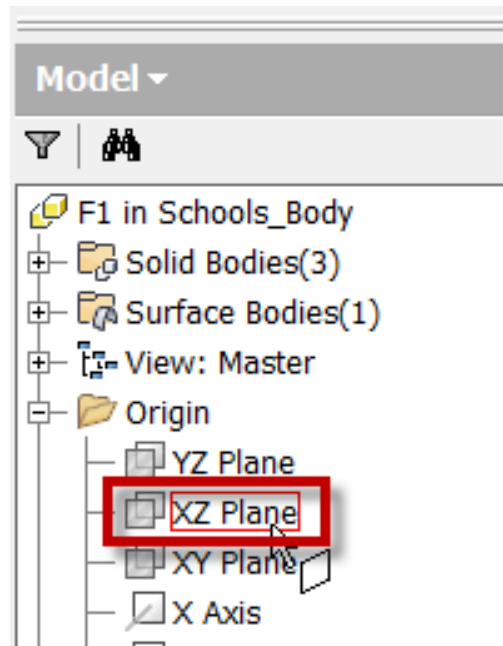
1. In the browser, right-click Form2 then click Solid Visibility.

**Note:** Your form number may be different; the rear wing should be selected.

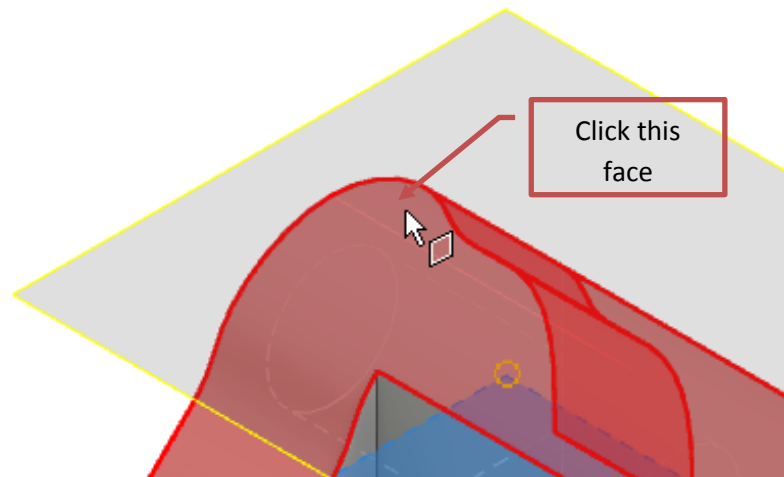


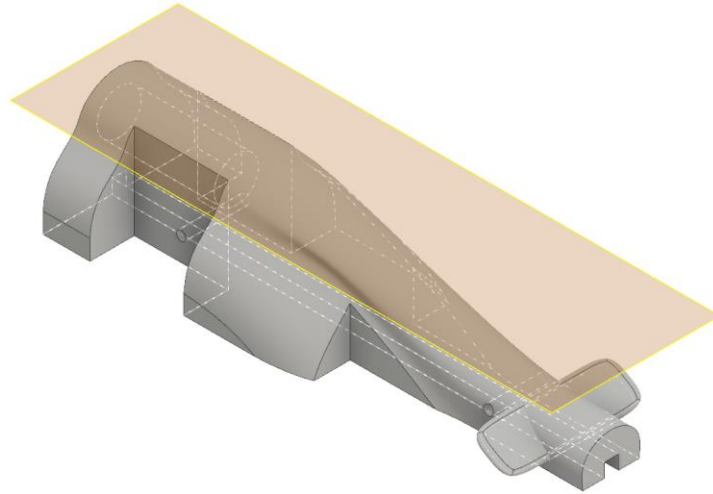
2. On the Work Features panel, click Plane.

3. In the browser, expand the Origin folder then click XZ Plane.

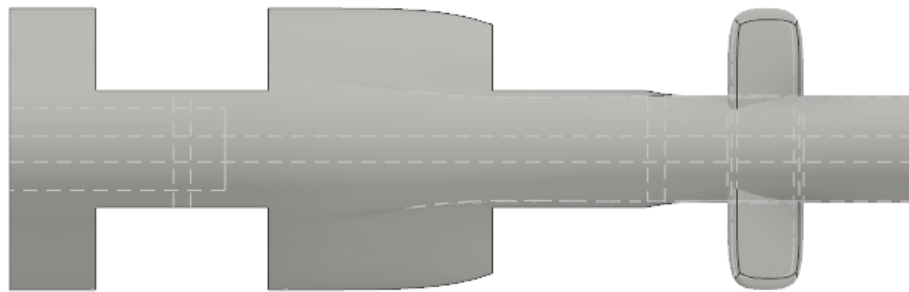


4. Click the top curved face at the rear of the car.





5. On the Sketch panel, click Start 2D Sketch.
6. In the graphics window, click the work plane.
7. Turn off the visibility of the work plane.
8. Rotate the block as shown.

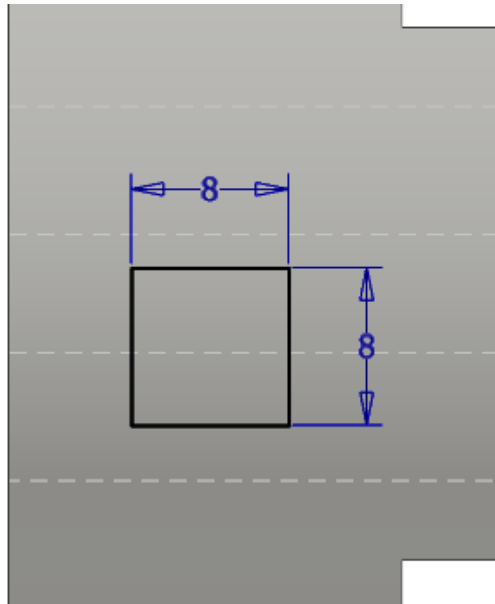


9. Zoom and pan to the left end of the block.



10. On the Create panel, click Rectangle.

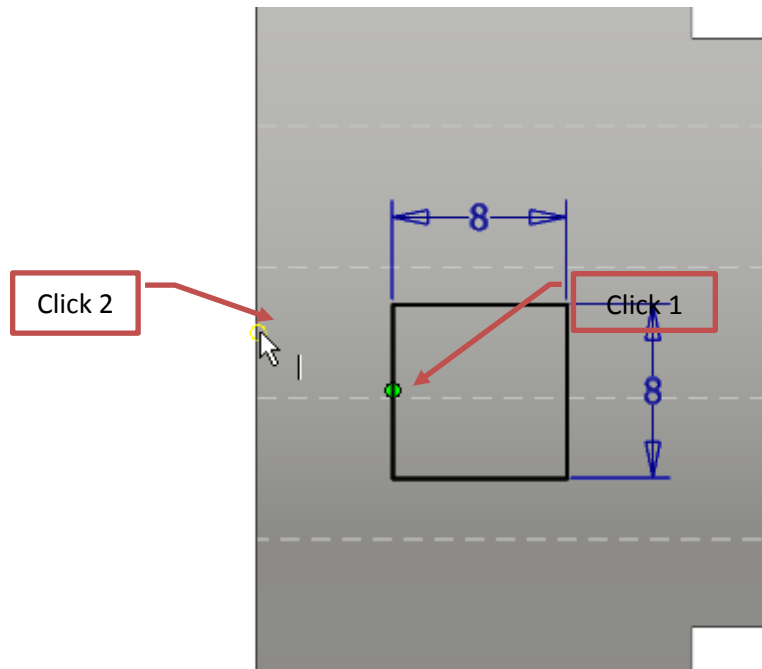
11. Create an **8mm x 8mm** rectangle.



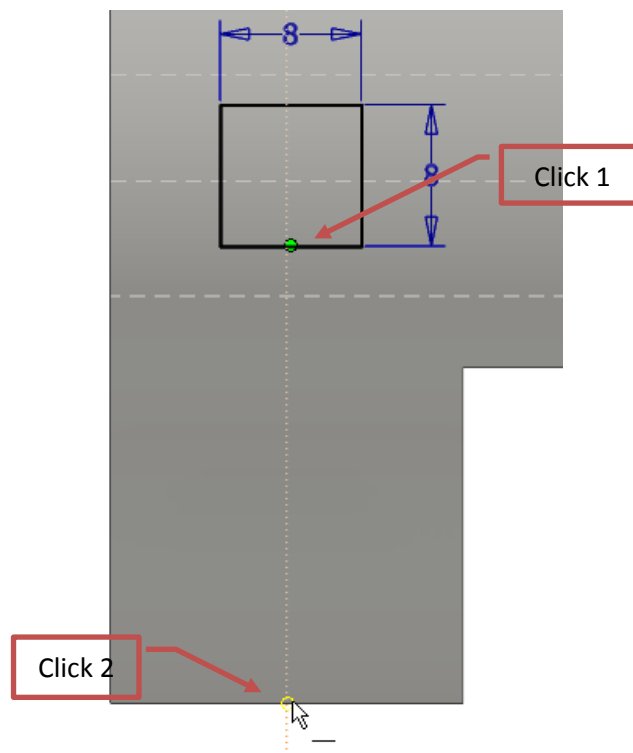
12. On the Constrain panel, click Vertical.

**Note:** You are applying a vertical constraint. It appears horizontal because the block is rotated about the axis of the sketch plane.

13. Click the midpoint on the left side of the rectangle (1) and the midpoint of the left edge of the block (2) located between the hidden edges of the tether slot.



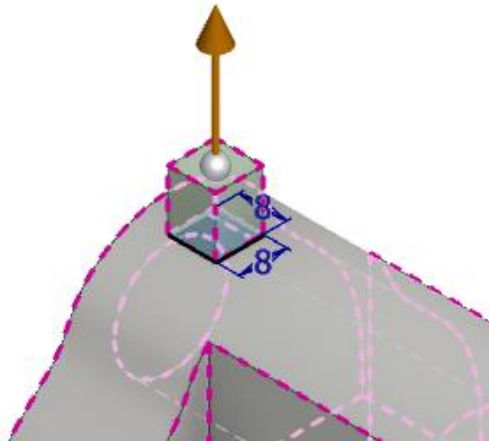
13. On the Constrain panel, click Horizontal.  
14. Create a constraint between the midpoint of the lower edge of the rectangle (1) and the bottom edge of the block (2).



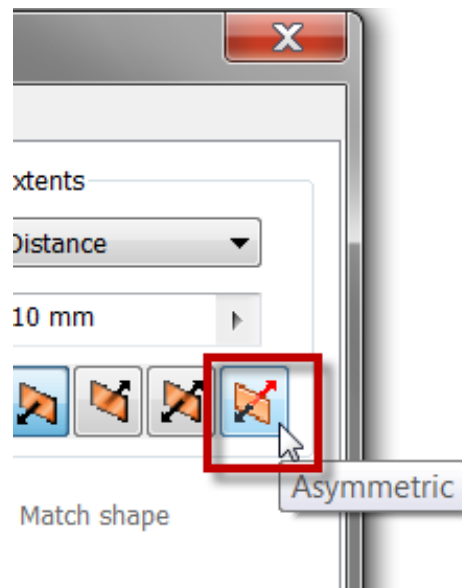


15. Press **E**.

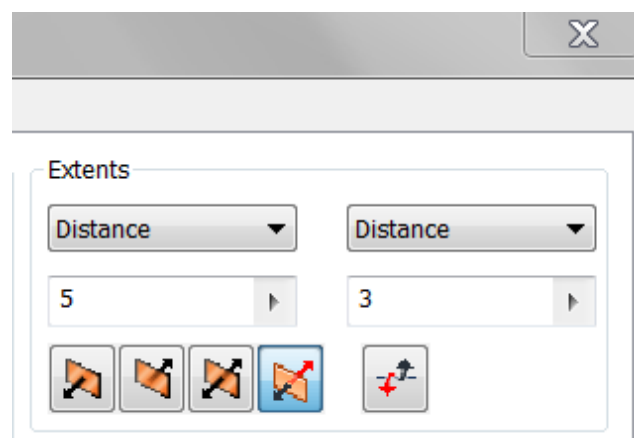
16. For the profile, the rectangle should be automatically selected.



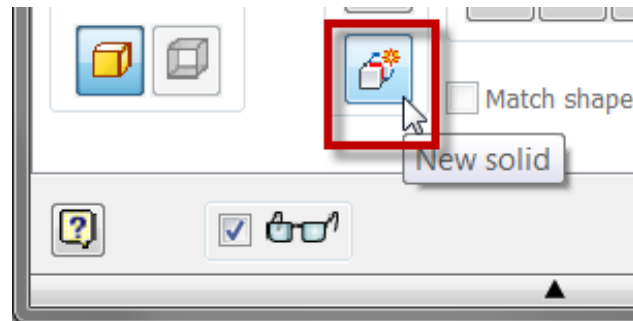
17. In the Extrude dialog box, click Asymmetric.



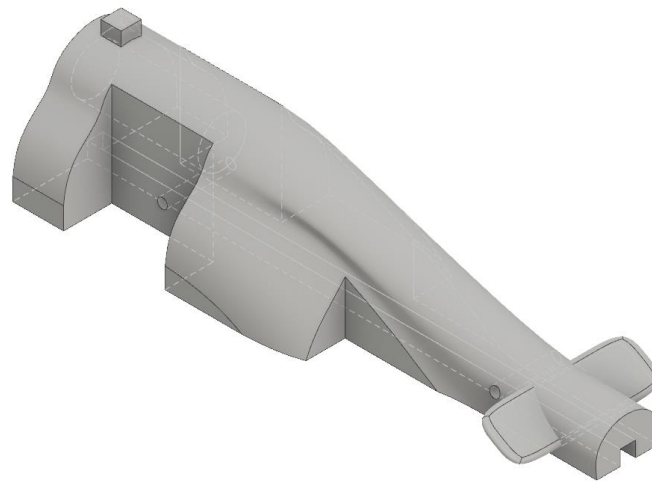
18. Under Extents, for distance 1 enter 5 then for distance 2 enter 3.



19. For Operation, select New Solid.



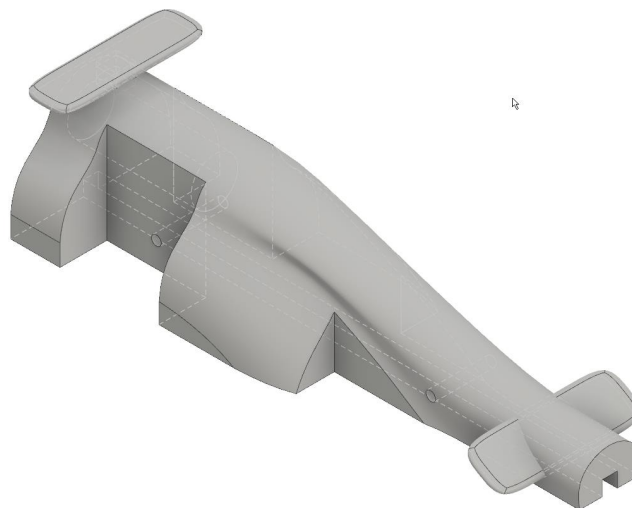
20. Click OK.



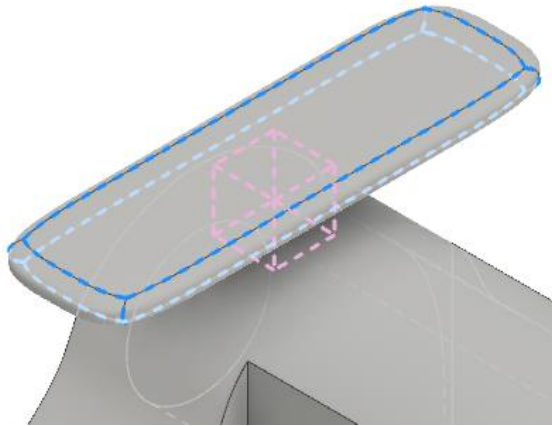
**Note:** A new solid is created so that it can be subtracted from the rear wing. The wing will be 3D printed so it requires a rectangular slot for assembly.

#### Step 4: Add the Wing Support on the Rear Wing

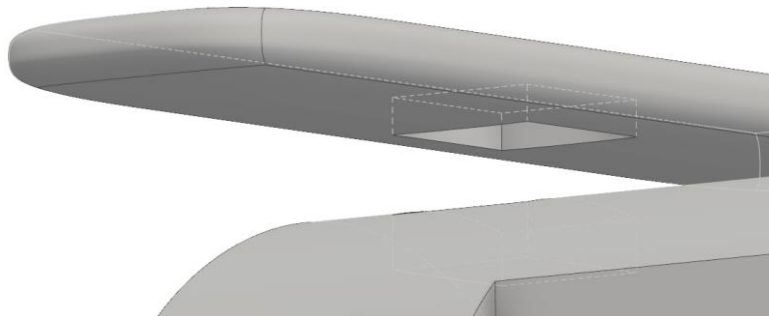
1. Turn on the visibility of Form2 (the rear wing).



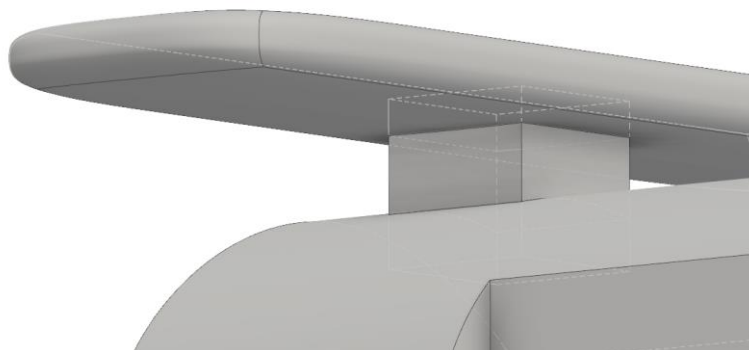
2. On the Modify panel, click Combine.
3. For Toolbody, select the rear wing.



4. For Base, select the extruded feature Solid4. It is easier to select it from the browser.
5. Click Keep Toolbody.
6. Under Operation, select Cut.
7. Click OK.
8. Rotate the block to view the cut in the rear wing.

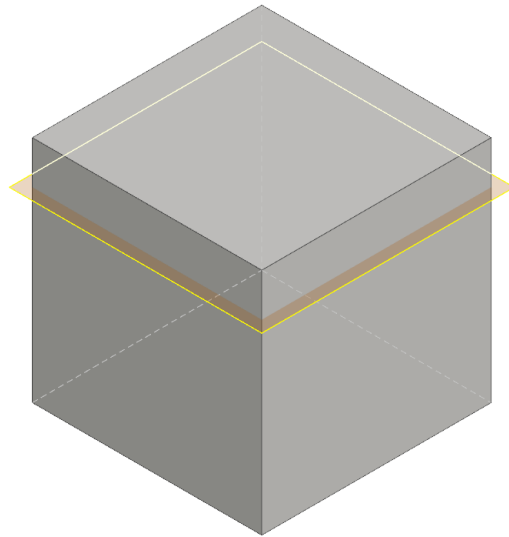


9. In the browser, under Solid Bodies, turn on the visibility of Solid4.

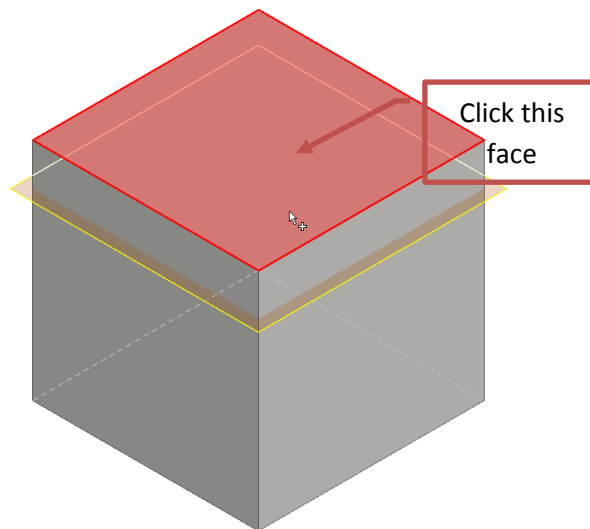


10. In the browser, right-click Solid4 then click Hide Others.
11. Zoom into the extruded feature.
12. On the Work Features panel, click Plane.

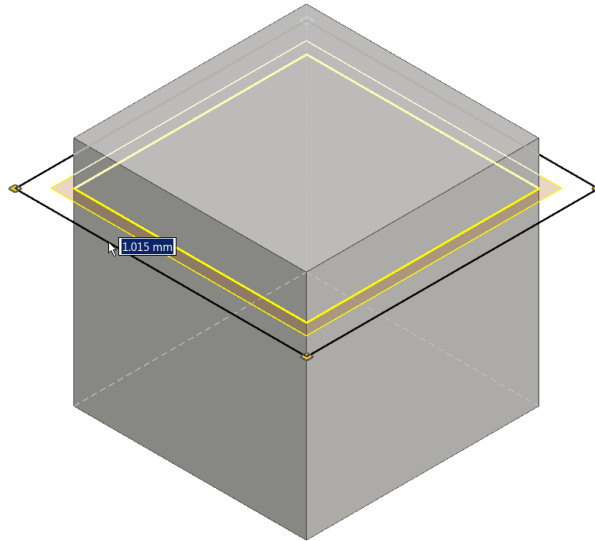
13. Select the top face of the body then drag it downwards. For distance, enter **-1.5**. Click OK.



14. Start a new 2D Sketch on the work plane.
15. Press F6 to return to the Home view. If required, zoom and pan to view the body.
16. On the Create menu, click Project Geometry.
17. Click the top face of the body.



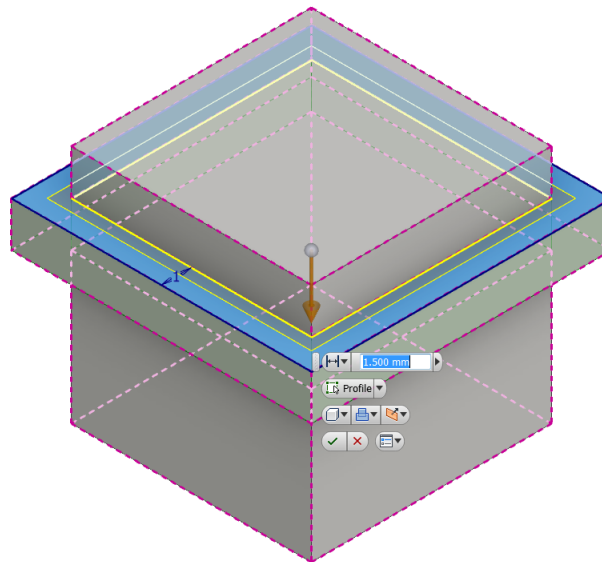
18. On the Modify panel, click Offset then drag the projected sketch away from the body.



19. For distance, enter **1** then press Enter.

20. Press **E**.

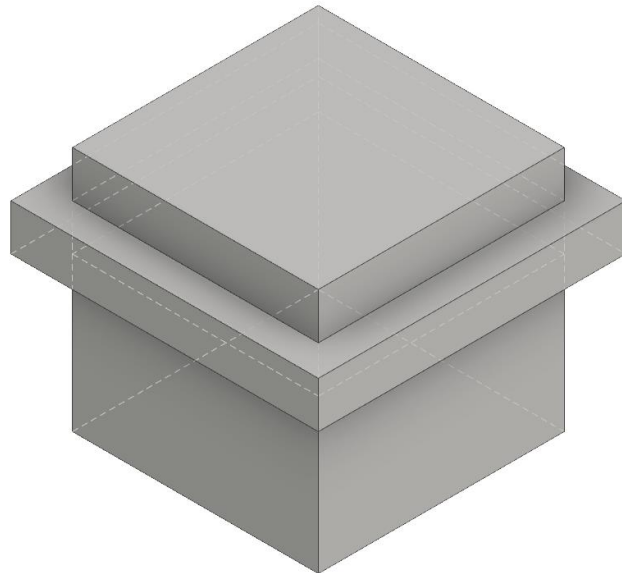
21. Select the profile around the body then drag the arrow on the preview to change the extrusion direction to down.



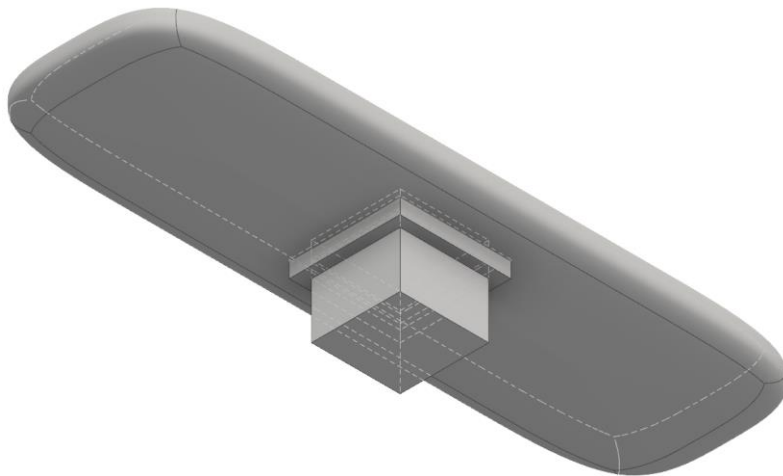
22. For Distance, enter **1.5**.

23. Select New Solid then click OK.

24. Turn off the visibility of the work plane.



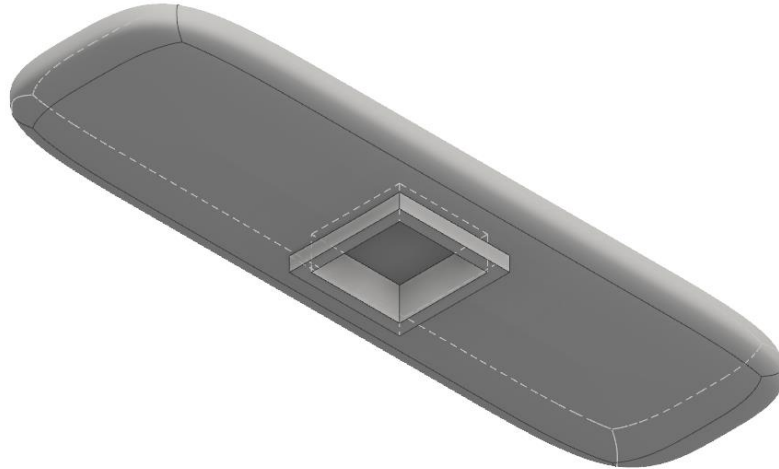
25. In the browser, right-click Solid2 (rear wing) then click Visibility. Rotate the part to view the underside.



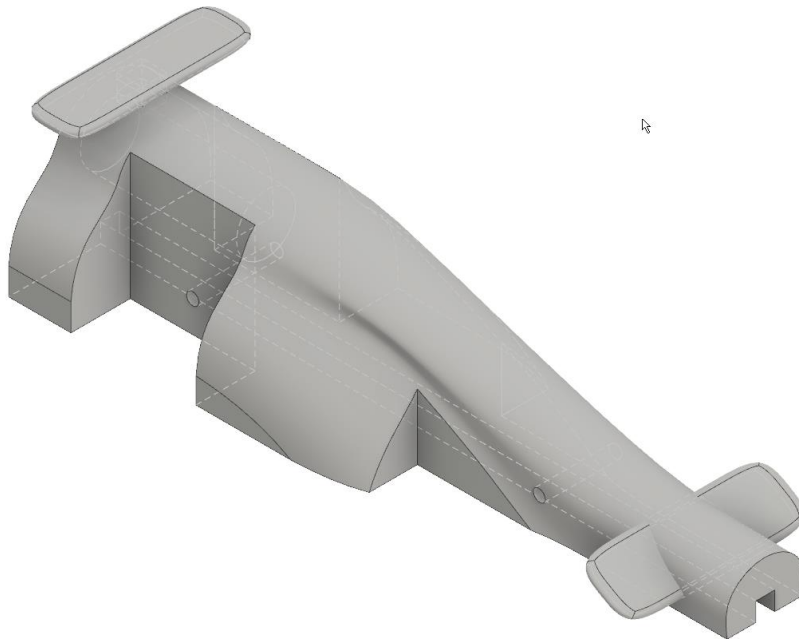
26. On the Modify panel, click Combine.

27. Repeat the previous workflow to join the rear wing Solid2 (Base) to the new extrusion Solid5 (Toolbody). In this workflow you are joining the bodies and not keeping the toolbody.

28. In the browser, turn off the visibility of Solid4. The rear wing (Solid2) now has a hole and support to assist in the assembly of the car. This body will be exported as an STL file and 3D printed.



29. In the browser, right-click Solid2 then click Show All.
30. Press F6 to return to the Home view.



29. Using the same workflow as the previous steps, combine the extruded feature (Solid4) and the main block (Solid1). In this workflow, you are joining the bodies and not keeping the toolbody.

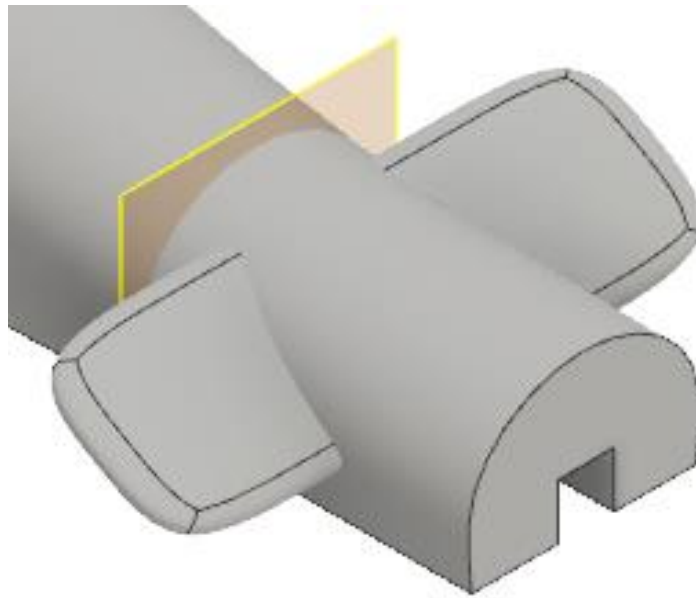
**Note:** The extruded feature is joined to the main block so that fillets can be added. This will be done later in the tutorial. The extruded feature and the block are machined as one piece and the fillets allow the 6mm router to machine around this feature.

31. Save the file.

## Step 5: Split the Body to Create a Nose Cone

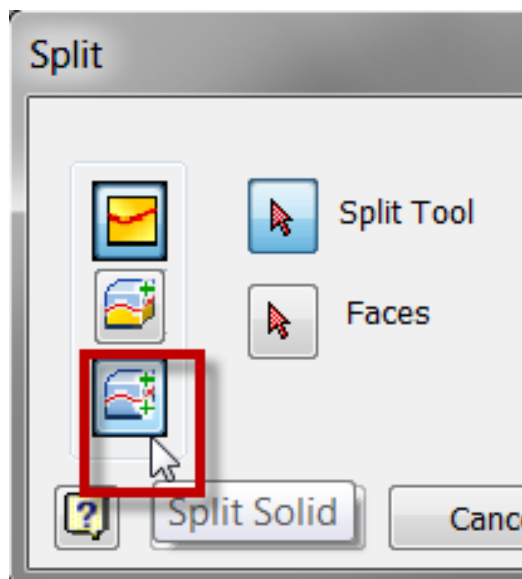
As part of the car design it has been decided that the nose cone should be a separate body. To do this, use the split tool to create two solid bodies.

1. On the Work Features panel, click Plane.
2. Select the front face of the car, then, drag the plane into the car. For distance, enter **-35**.



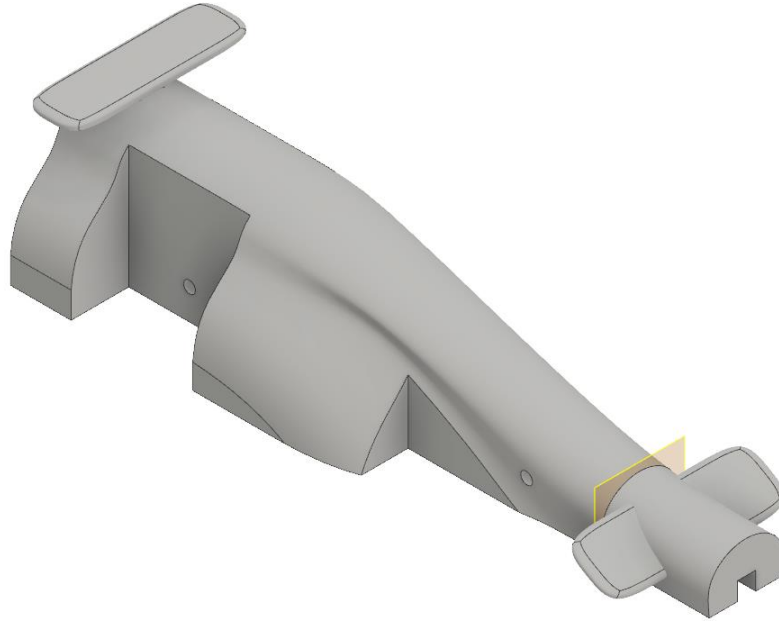
**Note:** If the front wing and the work plane interfere, you need to move the front wing forward. Use Edit Form to move the form (Form3 in this example) forward and away from the work plane as shown.

3. On the Modify panel, click Split.
4. In the Split dialog box, click Split Solid.

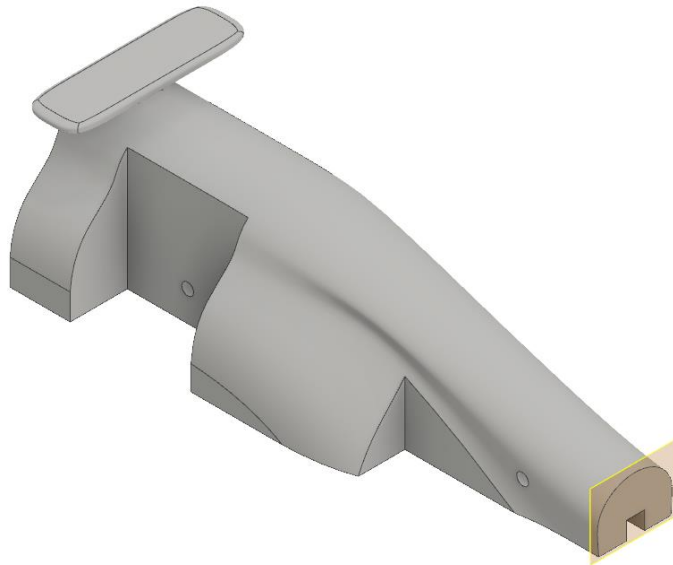




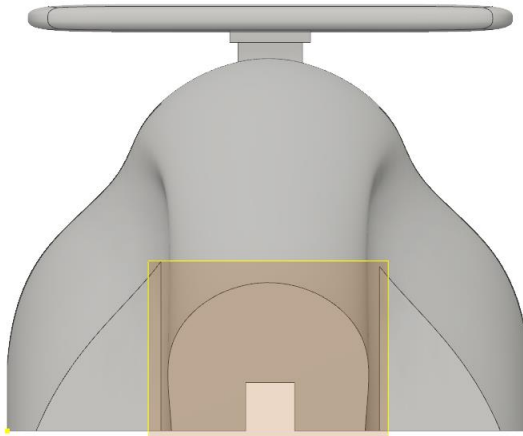
5. Click the work plane then click the car body
6. Click OK. The car body is split into 2 bodies.



7. In the browser, right-click Solid3 (front wing) then click Visibility. Repeat for Solid6 (front nose cone). Your numbers may be different.

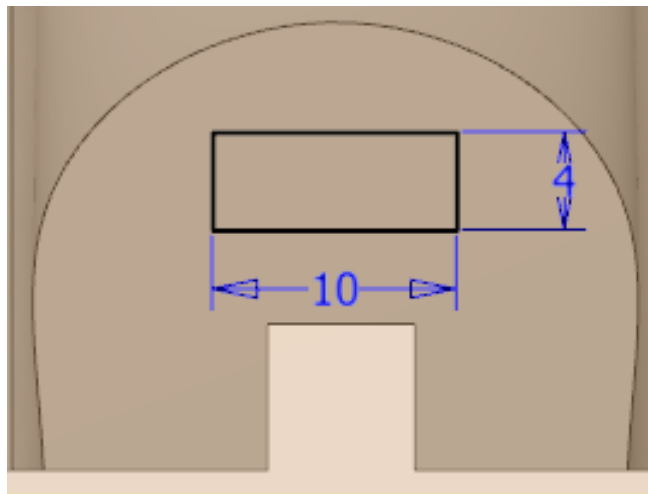


8. Create a new 2D Sketch on the work plane.

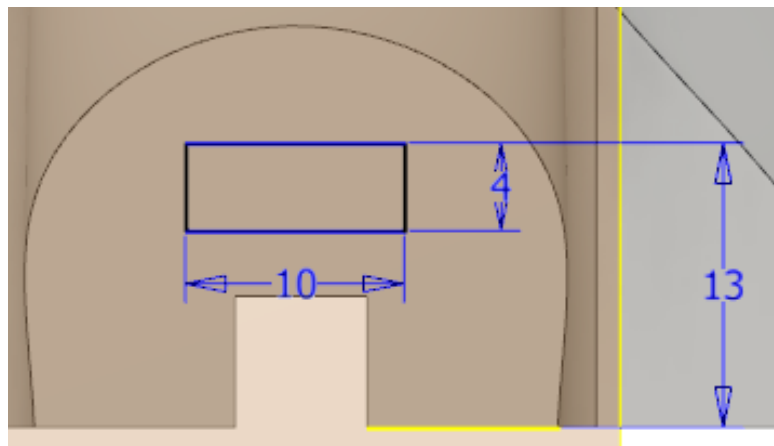


9. On the Create panel, click Rectangle.

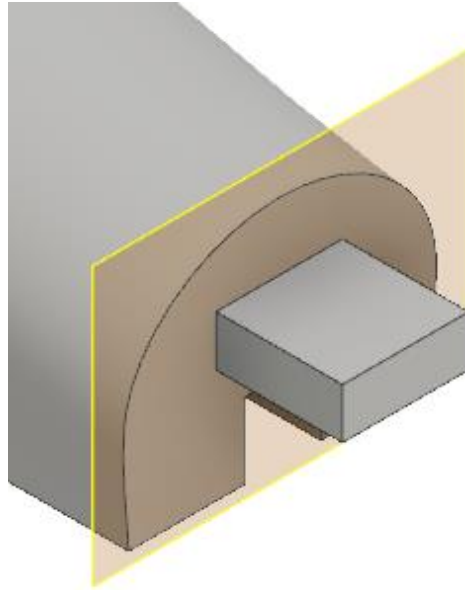
10. Sketch a **10mm x 4mm** rectangle as shown.



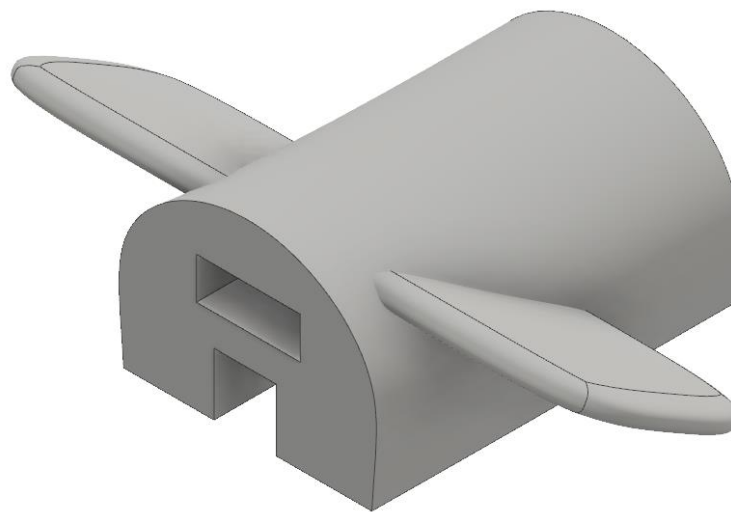
11. Add a **13mm** dimension between the top edge of the rectangle and the bottom edge of the car as shown.



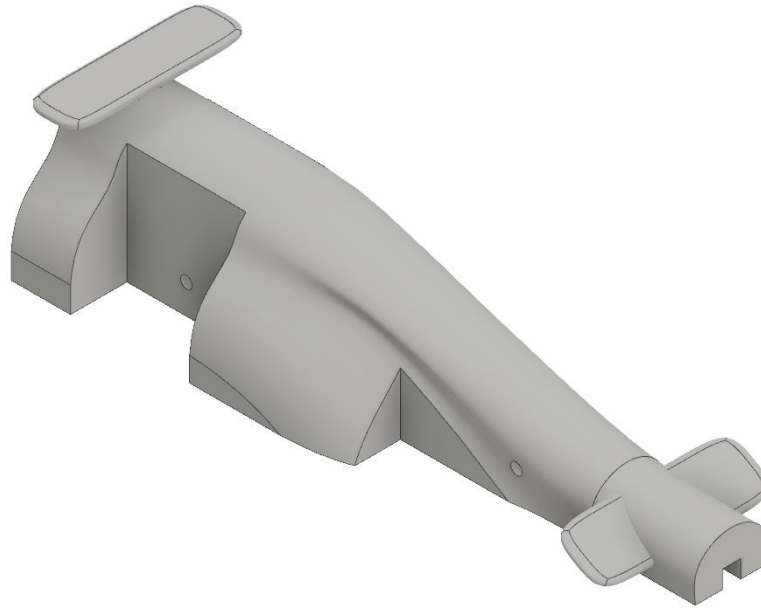
12. Add a vertical constraint to the rectangle and tether line slot to center the rectangle about the slot.
13. Press **E**.
14. Extrude the profile **10mm**.



15. Turn off the visibility of the work plane.
16. Turn on the visibility of Solid 6 and Solid7.
17. On the Modify panel, click Combine.
18. Cut Solid6 (Base) from Solid7 (Toolbody), making sure to keep the Toolbody. The slot is created in the nose cone of the car.
19. Review the nose cone and the slot.



20. Press F6 to return to the Home view and show all the bodies.

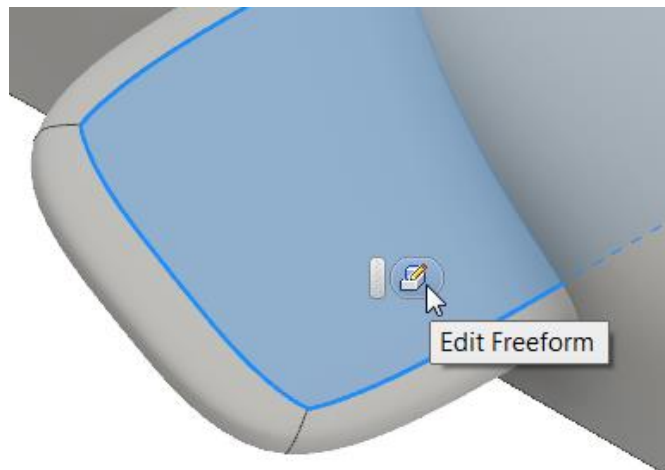


21. Save the file.

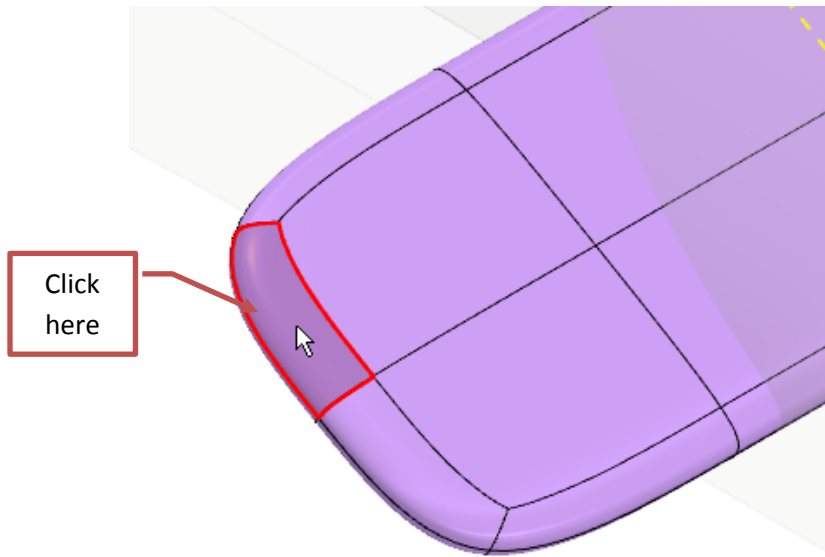
### Step 6: Editing the Wing Shapes

The wing shapes were created using a freeform box. The current shape can be modified to enhance its aerodynamic design. To do this, use the edit form tool.

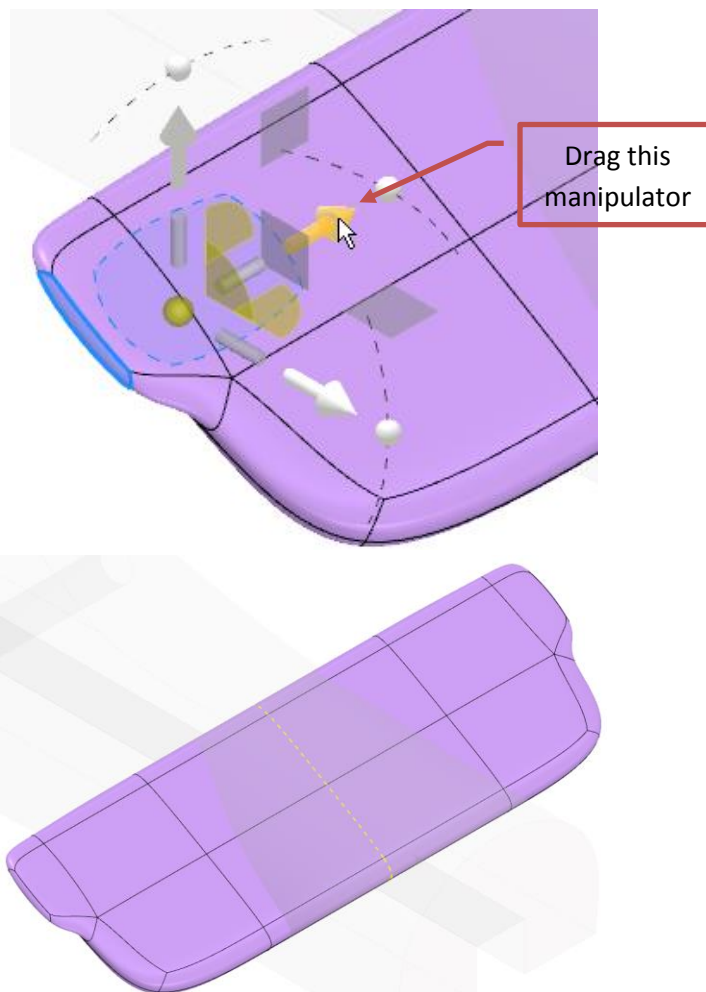
1. Click on the front wing then click the Edit Freeform icon.



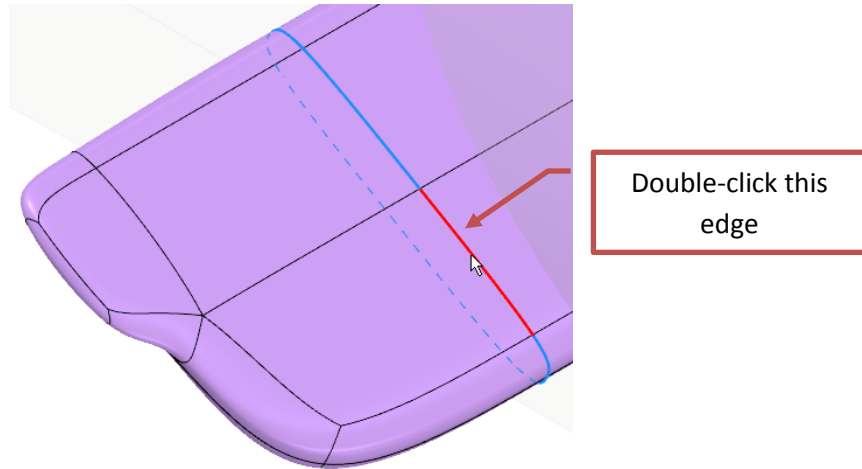
2. Click the face as shown.



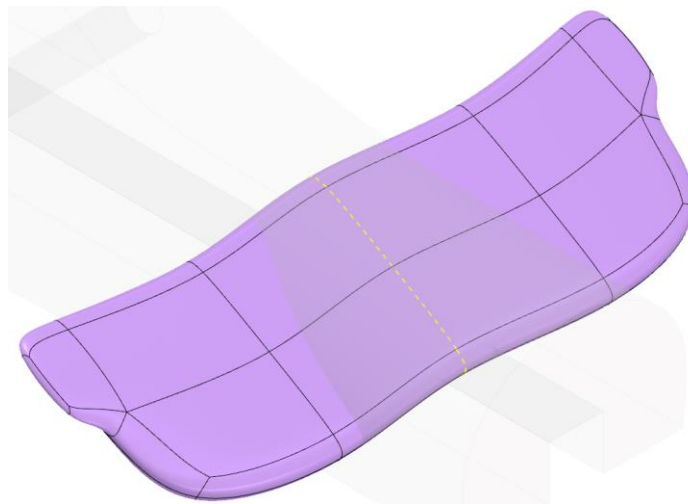
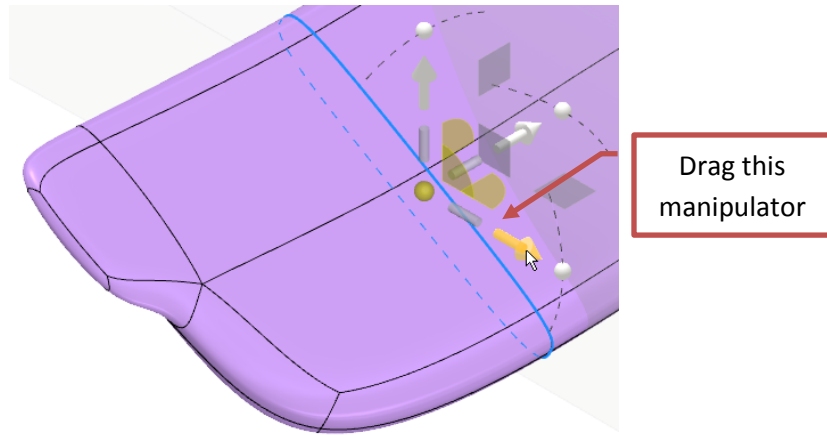
3. On the Edit panel, click Edit Form.
4. Hold down the Alt key then drag the manipulator **3mm** away from the car body.



5. Double-click the edge as shown. The complete edge around the wing should be selected.

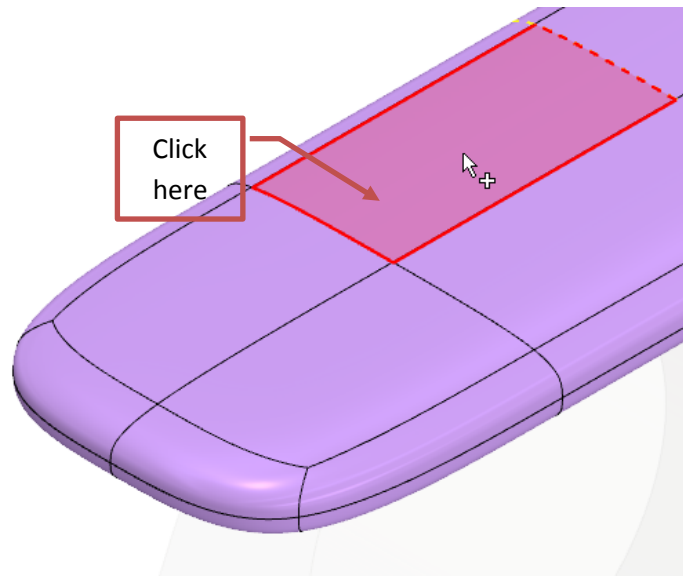


6. Drag the manipulator forward **4mm** as shown.

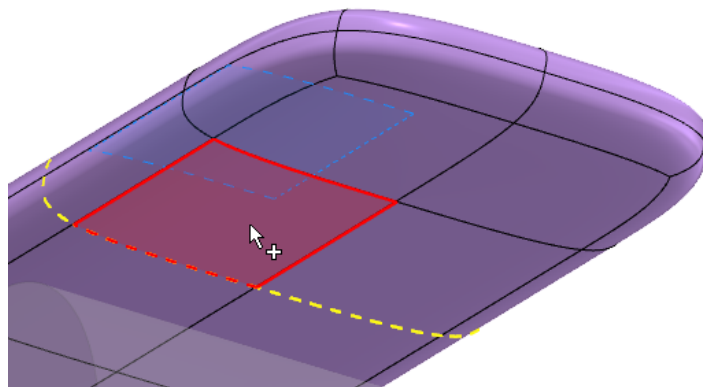


7. Click OK.
8. On the Exit panel click Finish Freeform.
9. Return to the Home view.

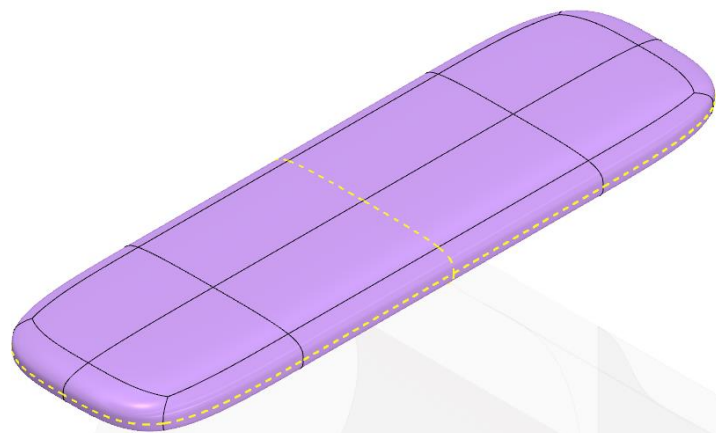
10. Click on the rear wing, then, click the Edit Freeform icon.
11. On the Symmetry panel, click Symmetry.
12. Click the face as shown.



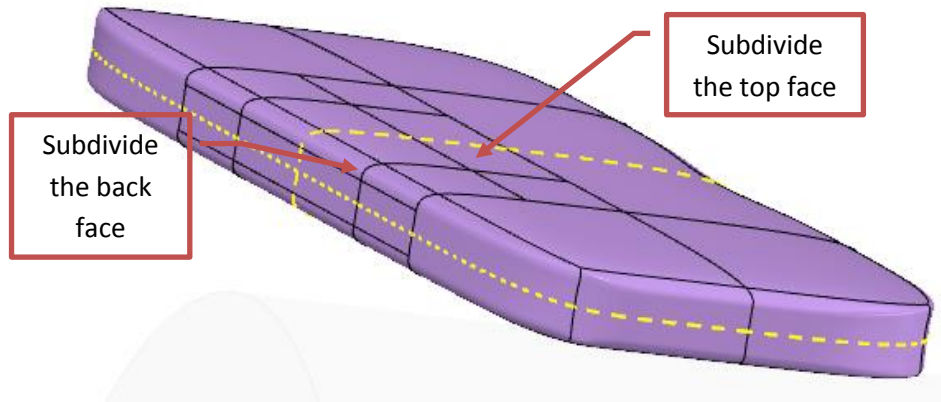
13. Rotate the wing, then, click the face directly underneath the previous selection.



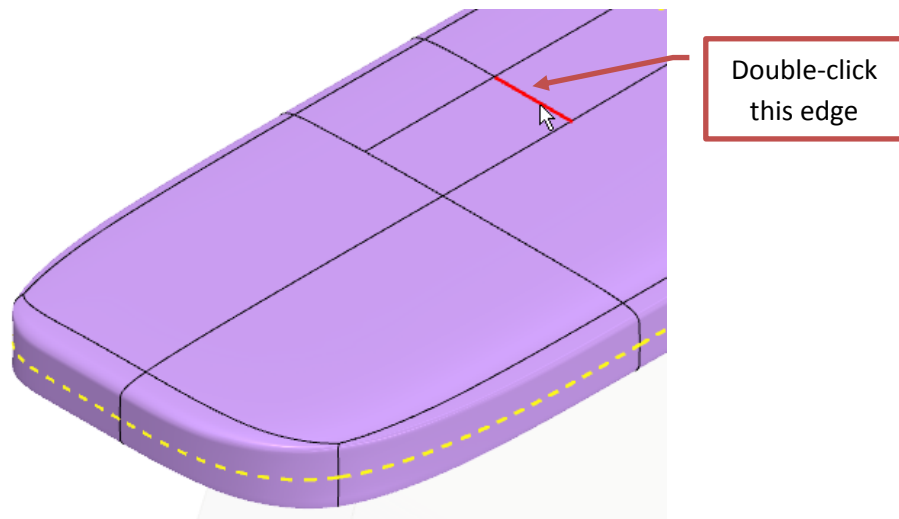
A second line of symmetry is added to the rear wing.



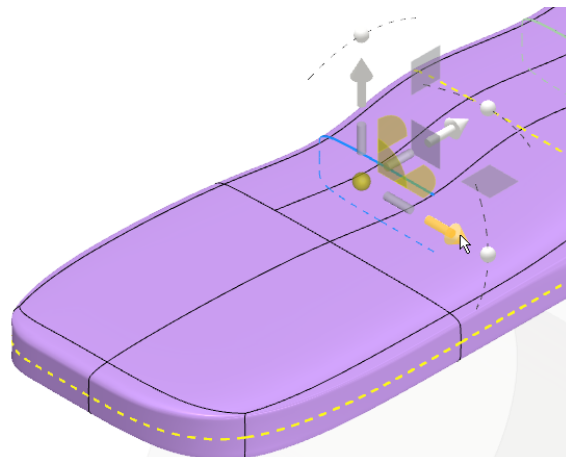
14. On the Modify panel, click Subdivide. Splitting the faces into smaller sections gives you more control over minor adjustments.
15. Click on the top face you selected previously, then rotate the wing and subdivide the back face as shown.



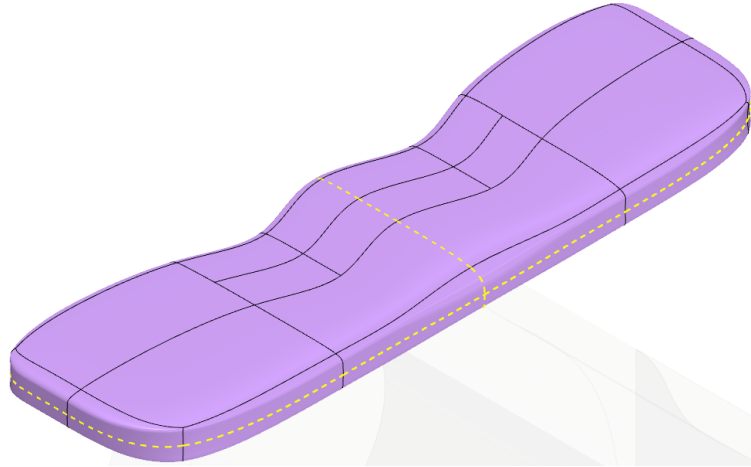
16. On the Edit panel click Edit Form.
17. Double-click the edge as shown.



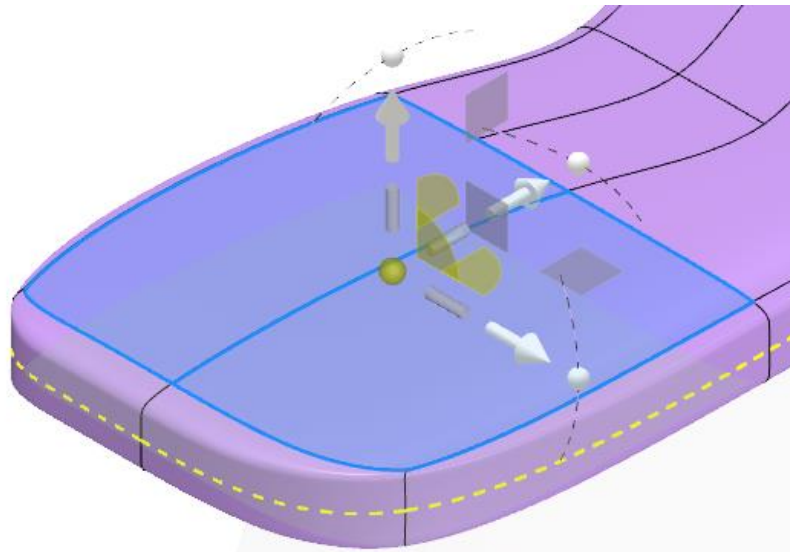
18. Drag the manipulator towards the front of the car **5mm**.



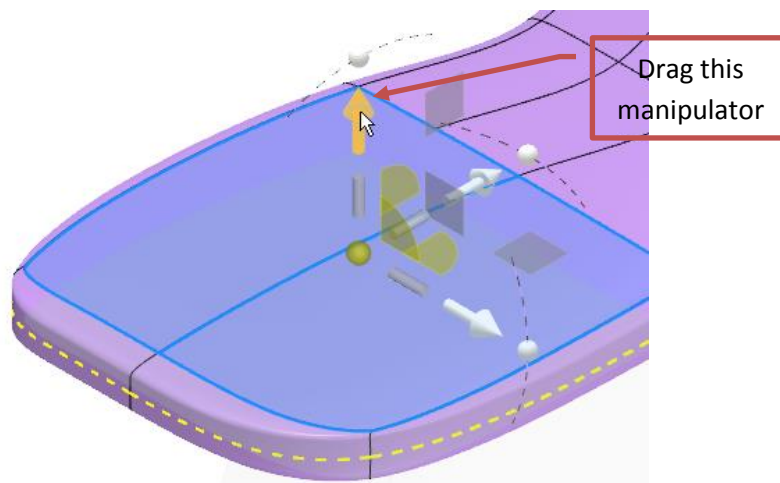


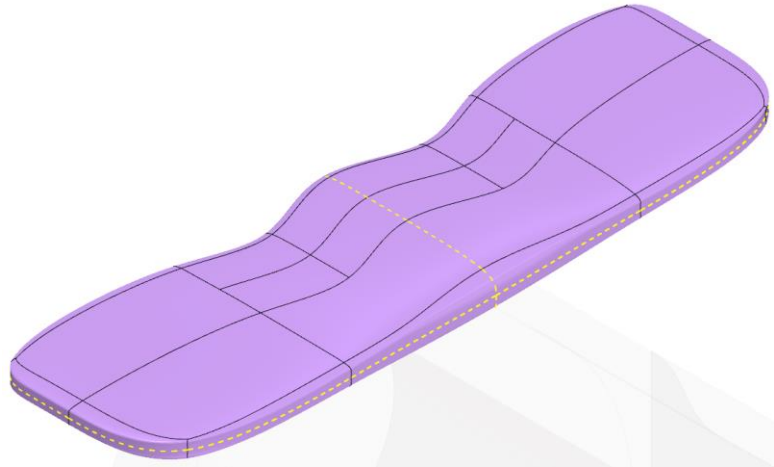


19. On the Edit panel click Edit Form.
20. Select the 2 large faces at the end of the wing, using Shift for multiple selections.



21. Drag the vertical manipulator down **0.8mm**.



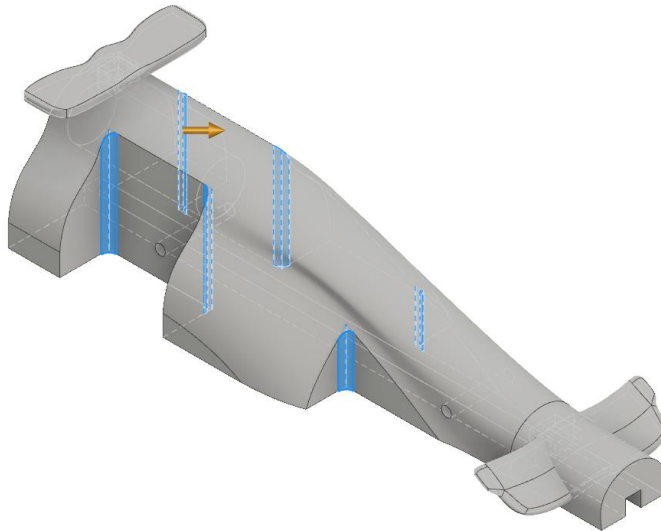


22. Click OK. Then, on the Exit panel, click Finish Freeform.
23. Return to the Home view.

### Activity 5: Adding Fillets to the Car

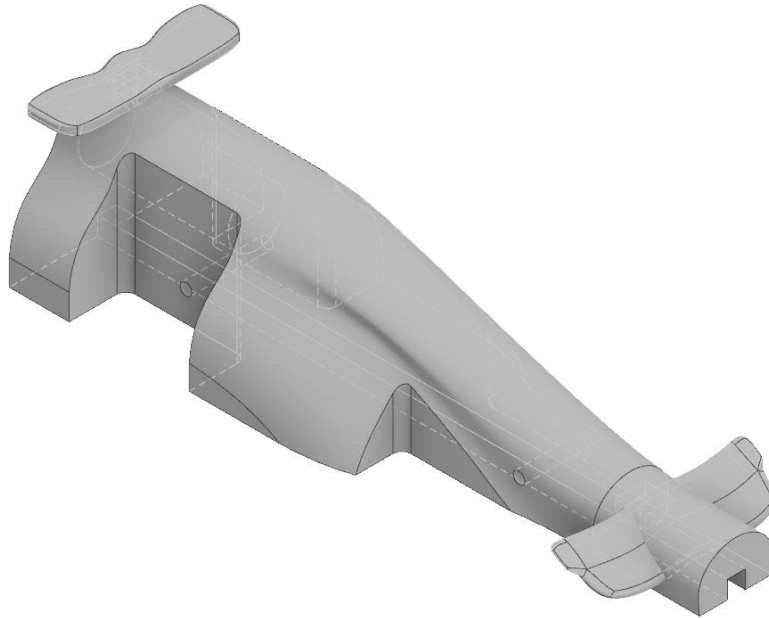
In this activity, you add fillets (rounded edges) to your car to assist with aerodynamics and to make it suitable for machining. For video instructional support, download the step-by-step video tutorials for this activity.

1. On the Modify menu, click Fillet.
2. Select the inside edges of the wheel surrounds on both sides of the car as shown.

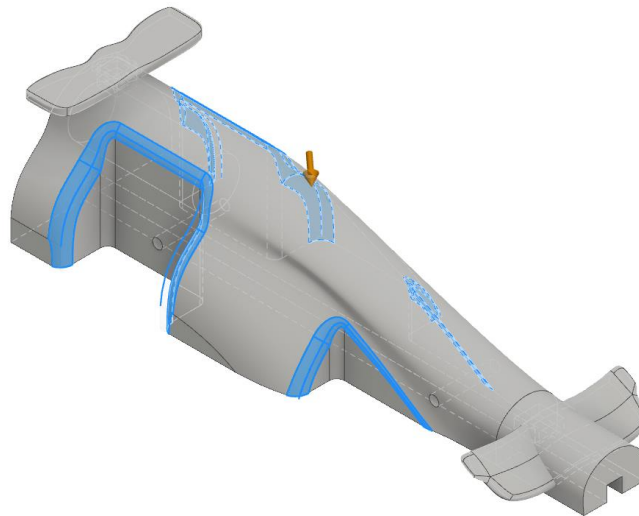


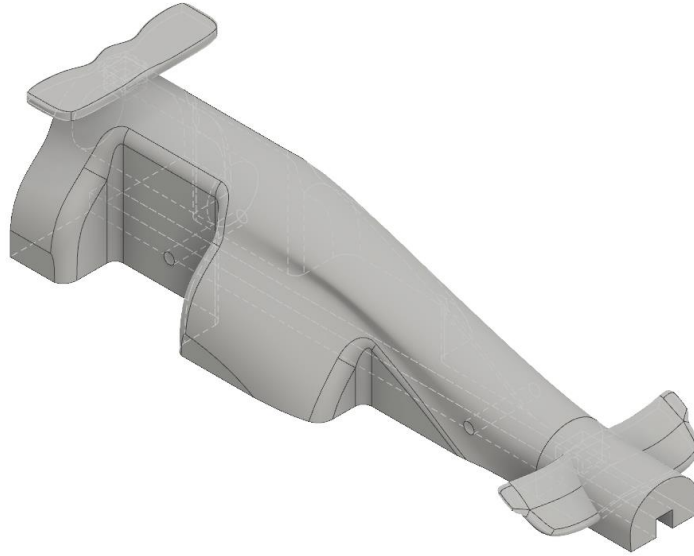
3. For radius, enter **3**. The cutting tool used to cut your car has a diameter of  $\frac{1}{4}$  inch/6.35mm, so it will leave a radius of 3mm on any internal cut (or 3.175mm if you want to be precise!).

4. Click OK.

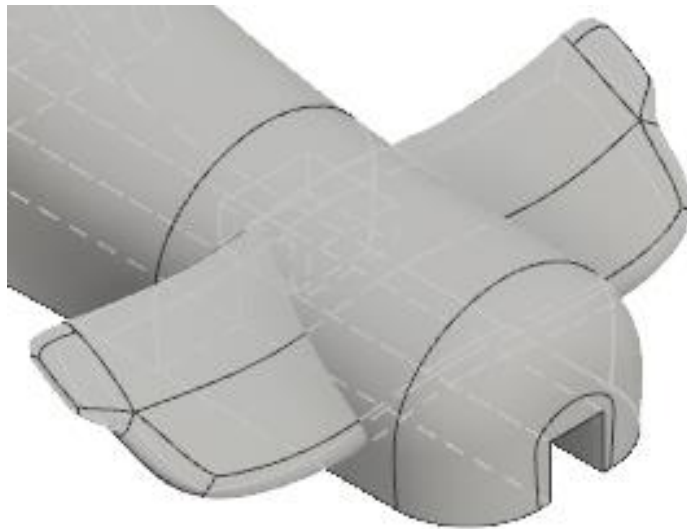


5. Add **5mm** fillets to the 4 edges as shown.

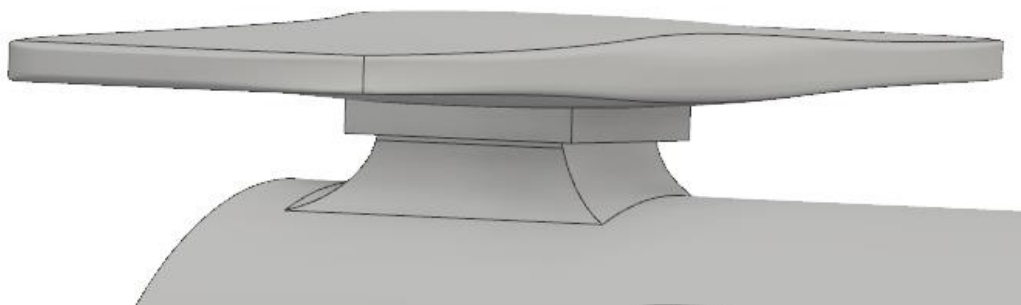




6. Add an **8mm** fillet to the nose cone as shown.



7. Add **3mm** fillets to the edges around the extrusion as shown.

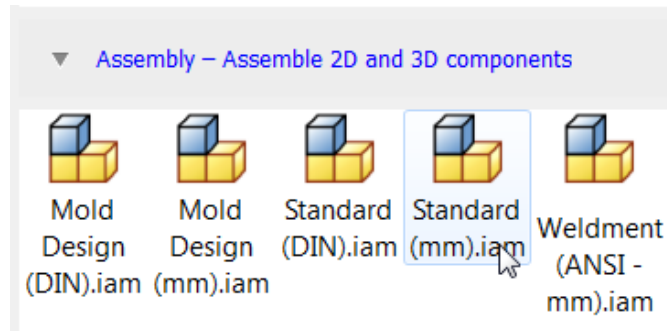


## Activity 6: Assemble the F1 in Schools Car

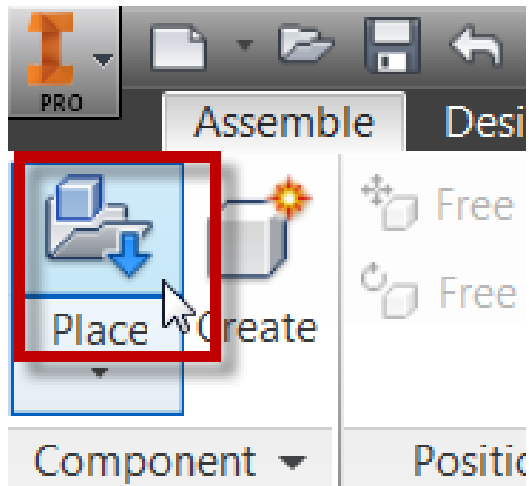
In this activity, you assemble the components that make up the complete car. You then check to make sure your Car Body design fits within the block. For video instructional support, download the step-by-step video tutorials for this activity.

### Step 1: Start a New Assembly File

1. On the Application menu, click New.
2. Under Templates, click Metric.
3. Under Assembly, click Standard (mm).

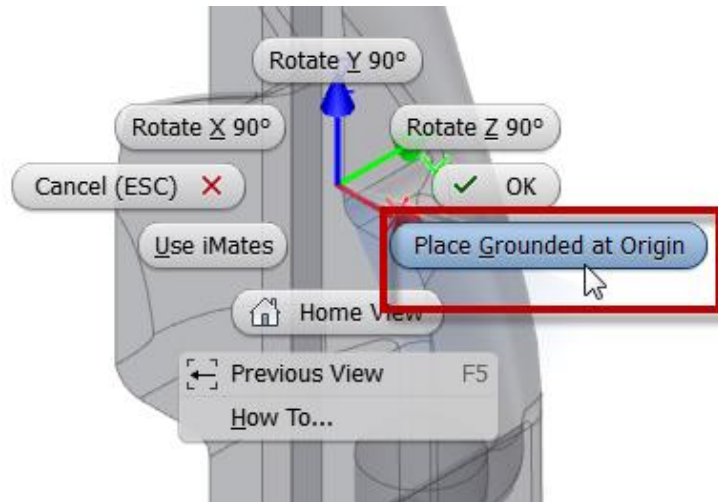


4. Click Create.
5. Save the file as **my F1 in Schools Car Assembly**.
6. On the Component panel, click Place.

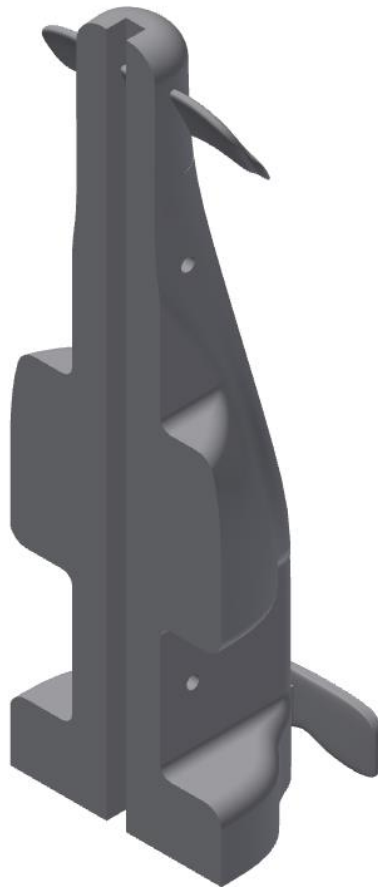


7. In the Place Components dialog box, select Inventor F1 in Schools Body then click Open. The selected component is placed in the graphics window attached to the cursor.

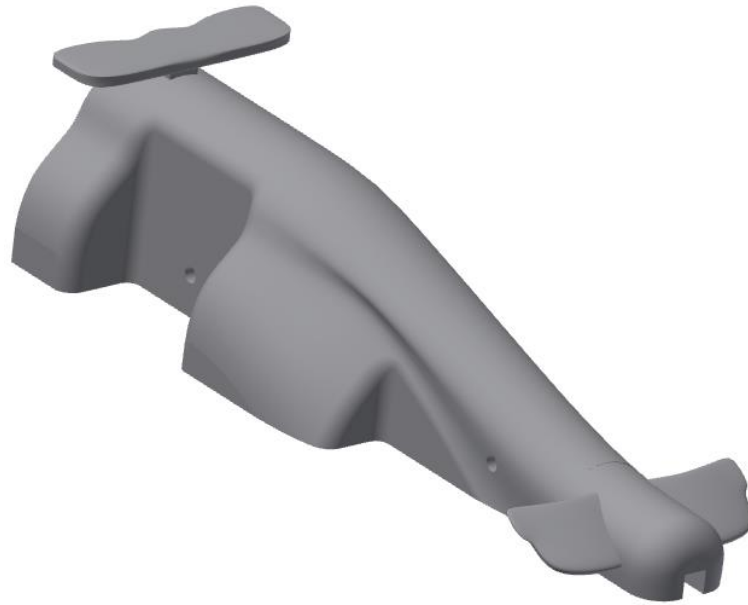
8. In the graphics window, right-click, then click Place Grounded at Origin.



9. Right-click, then click OK. You now set a different Home view.



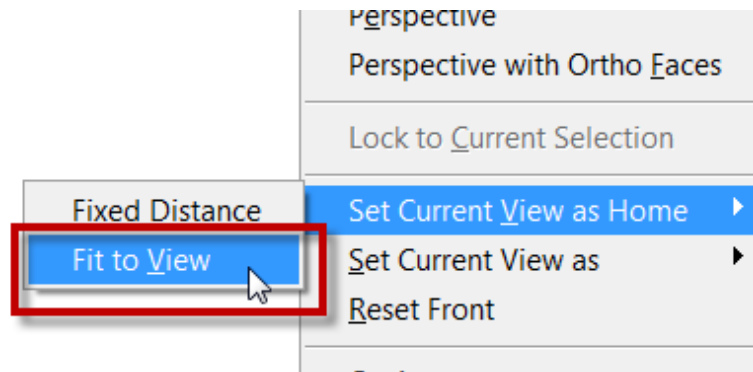
10. Rotate the car body approximately as shown.



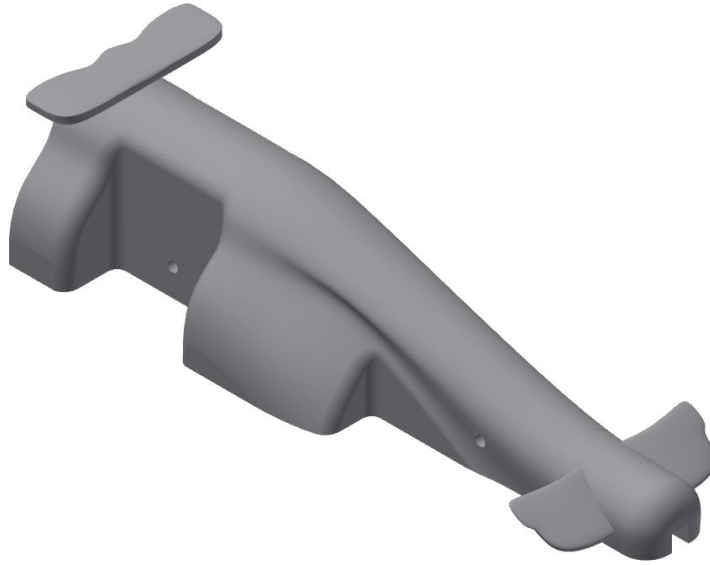
11. On the ViewCube, click the front corner as shown.



12. On the ViewCube, right-click, then click Set Current View as Home > Fit to View.

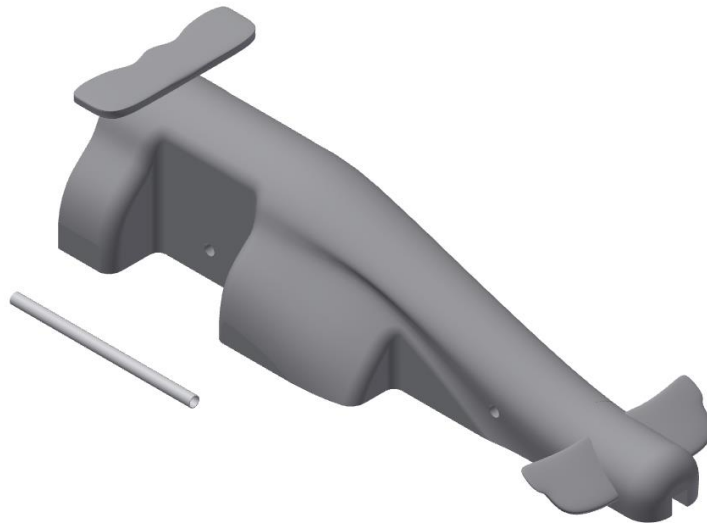


13. Press F6 to return to the Home view.



## Step 2: Place the Straws in the Assembly

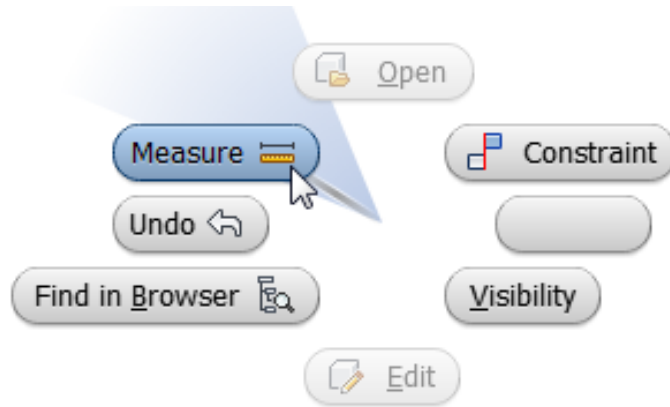
1. On the Component panel, click Place.
2. In the Place Components dialog box, select Inventor F1 in Schools Straw\_Rear then click Open.
3. Click to place the straw then right-click and click OK.



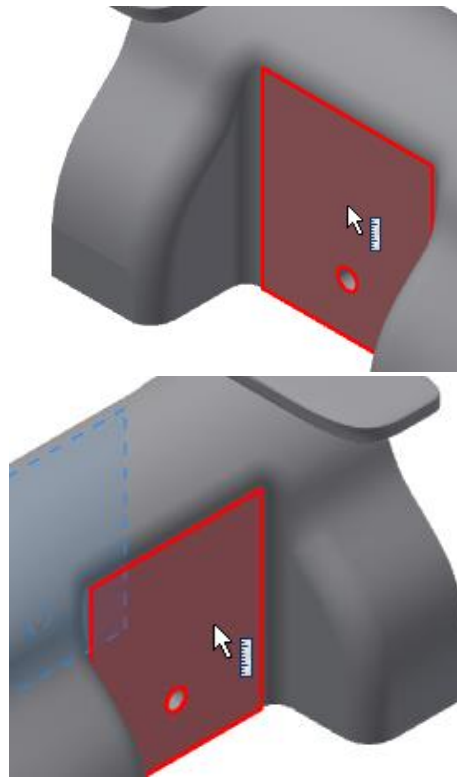
**Note:** The straw should be flush to the sides of the body at the rear and front of the car. You measure the distance to determine the length for each straw.



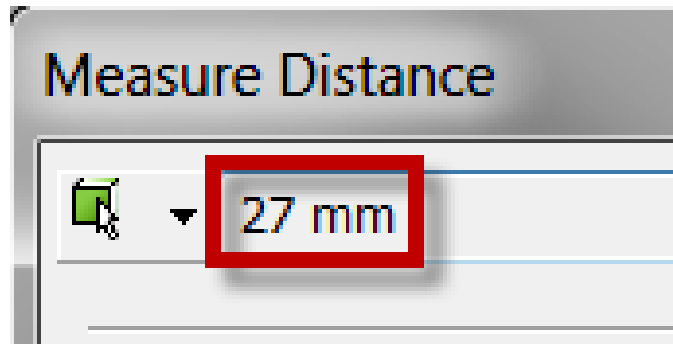
4. In the graphics window, right-click, then click Measure.



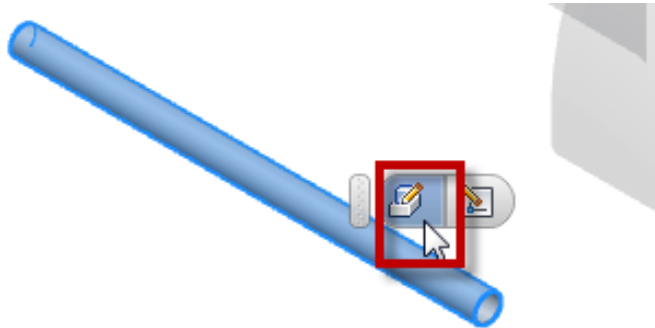
5. Select the front face of the wheel opening, then rotate the car body and select the opposite face.



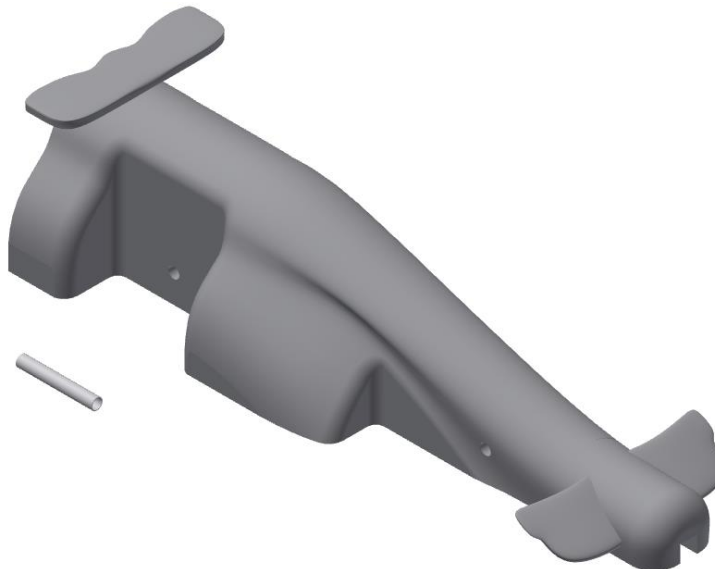
- Review the Measure dialog box. The distance is 27mm.



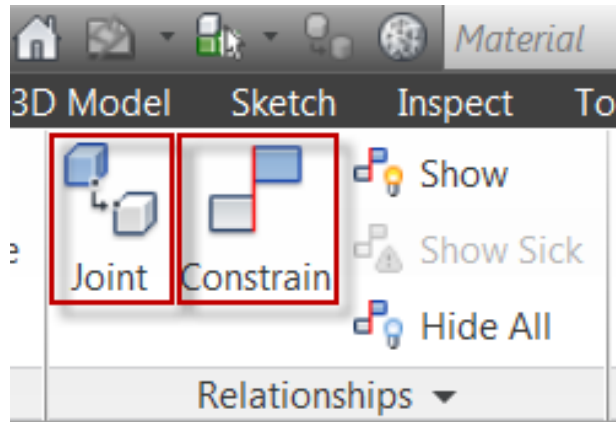
- Right-click, then click OK. Return to the Home view.
- In the graphics window, right-click the straw then click Edit.
- Click the straw then click Edit Extrude.



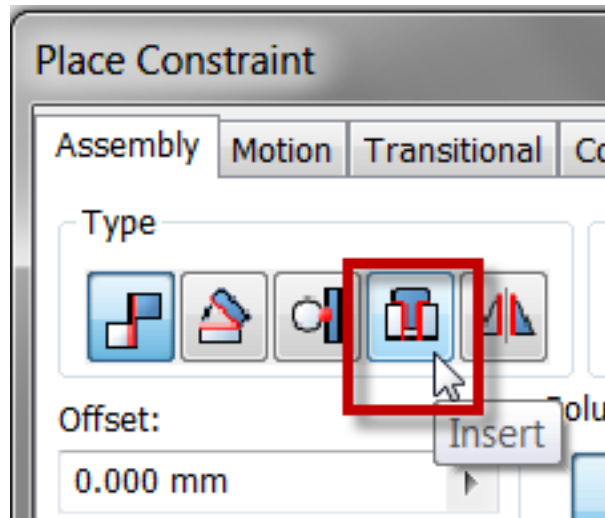
- For distance, enter **27** then click OK.
- On the Return panel, click Return.



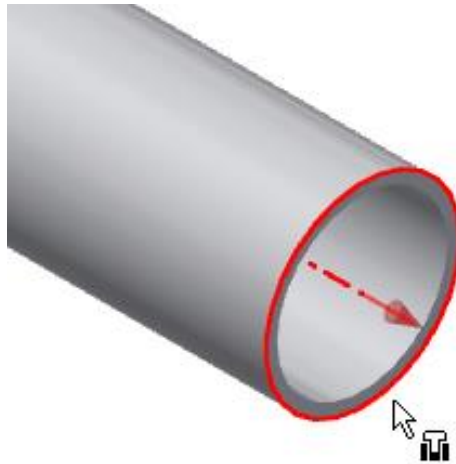
**Note:** There are 3 options for assembling the parts: Joint, Constrain and Assemble. In this tutorial, the focus is on Joint and Constrain. As each workflow is used, you can better determine when to use which tool.



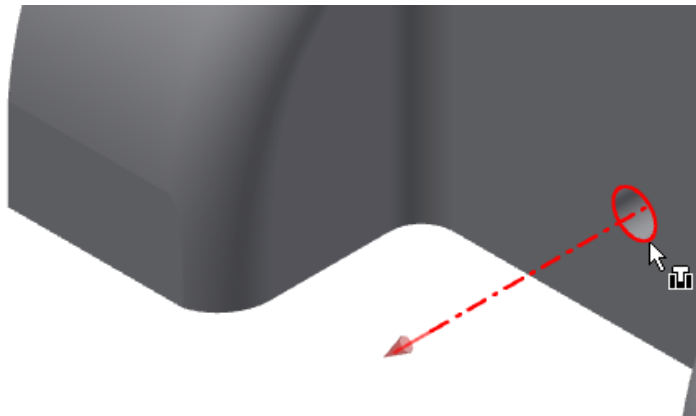
12. On the Relationships panel, click Constrain.
13. In the Place Constraint dialog box, click Insert.



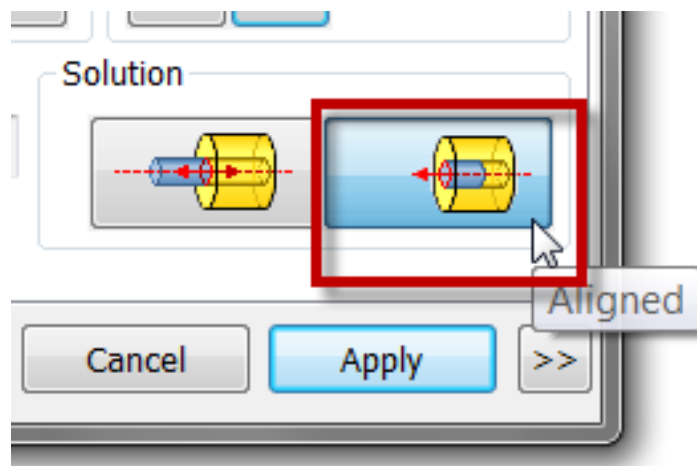
14. Select the outside edge of the straw as shown.



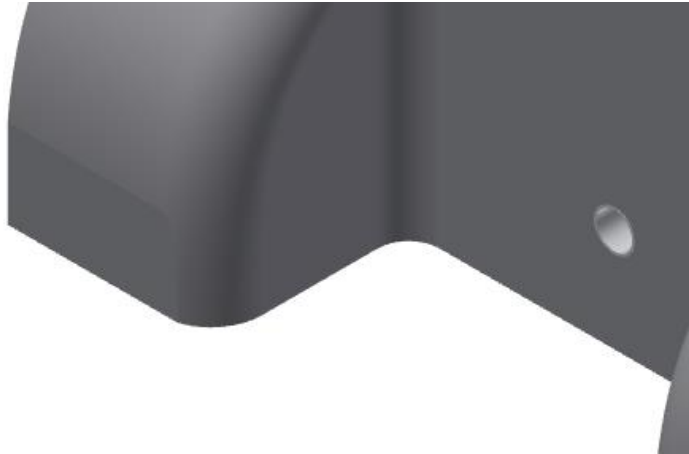
15. On the car body, select the edge of the hole.



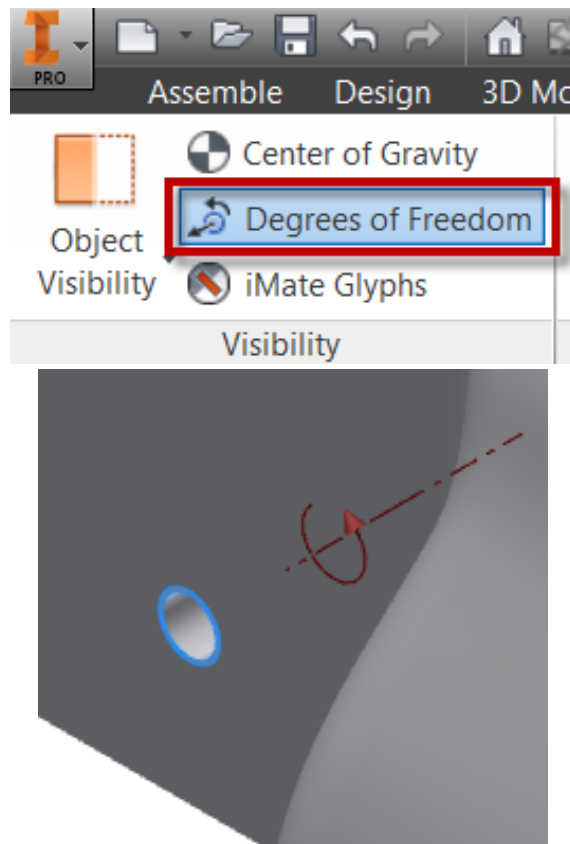
16. In the Dialog box, click Aligned.



17. Click OK.

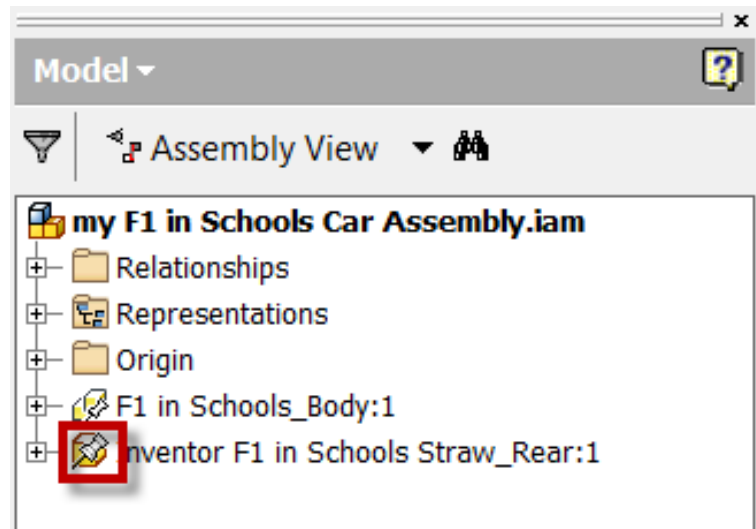


18. On the View tab > Visibility panel, click Degrees of Freedom. Note that the straw has one degree of freedom: rotational about the X-axis.



**Note:** The straw should have no degrees of freedom. The easiest workflow is to ground the part now that it is in the correct position.

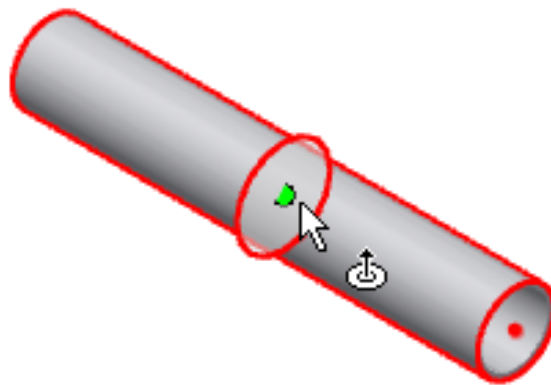
19. In the browser, right-click the straw, then click Grounded. The degrees of freedom icon is no longer displayed.



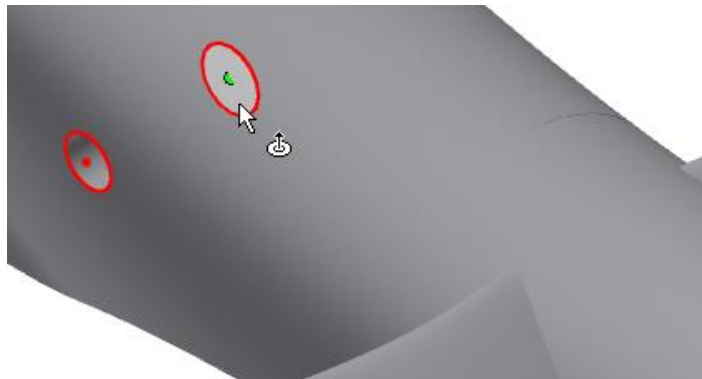
20. On the View tab > Visibility panel, click Degrees of Freedom to turn off the display.
21. Repeat the workflow to place, measure, and edit the Inventor F1 in Schools Straw\_Front part.

**Note:** The curved surface around the hole at the front of the car means you cannot use the insert constraint. You will use the Joint tool.

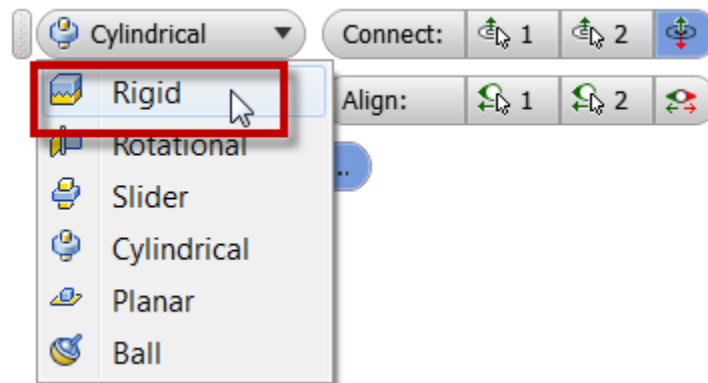
22. On the Relationships panel, click Joint.
23. Move the cursor over the straw, then click when the center is highlighted as shown.



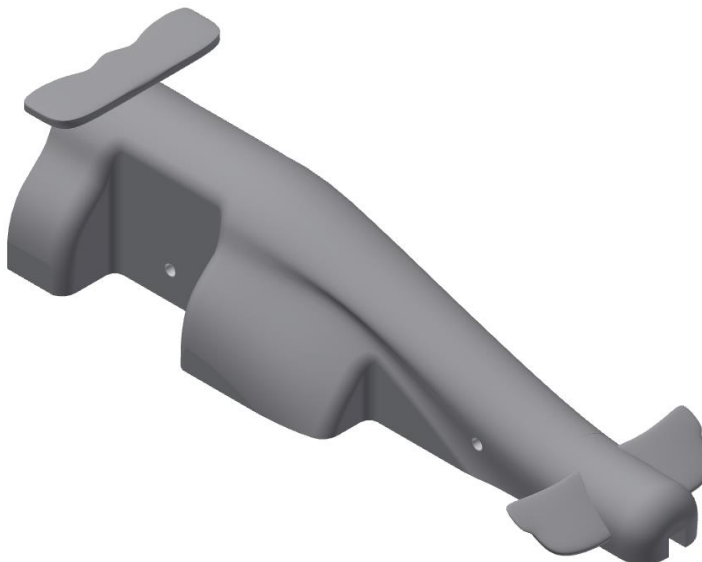
24. Move the cursor over the hole in the car body; click when the center is highlighted as shown.



25. From the joints type list, select Rigid. This will remove all degrees of freedom.



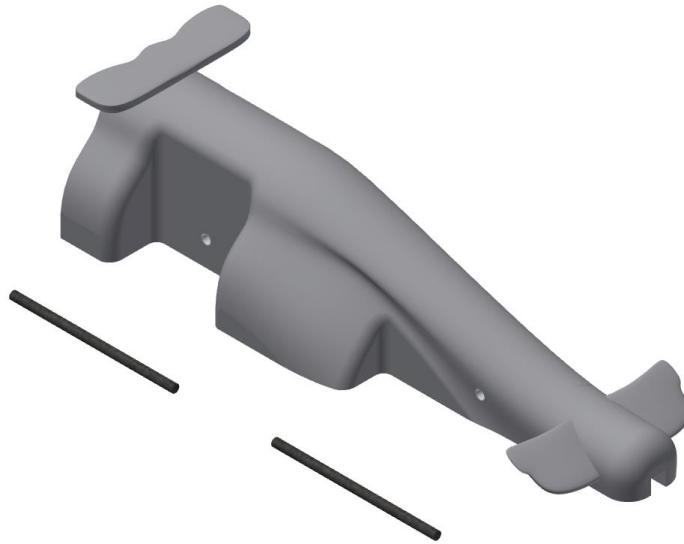
26. Click OK.



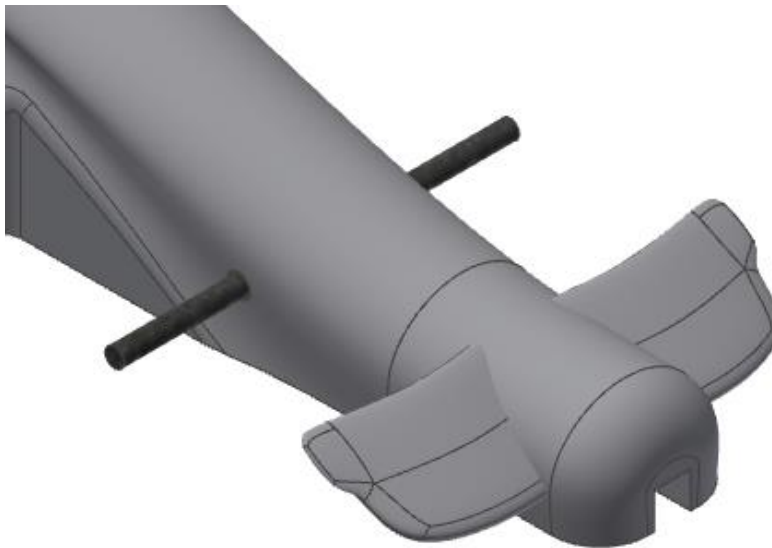
27. Save the file.

### Step 3: Place the Axles in the Assembly

1. Place two Inventor F1 in Schools Axle parts in the assembly.

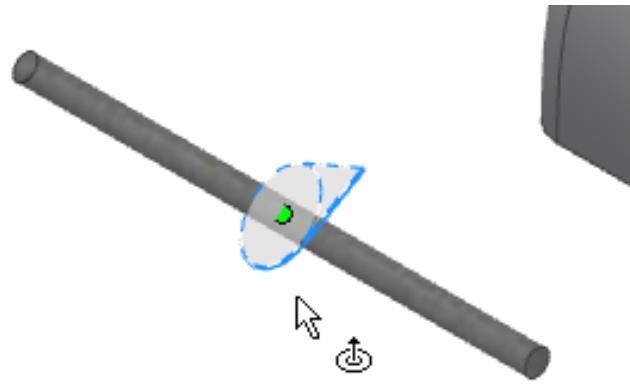


2. On the Relationships panel, click Joint.
3. Using the workflow from the previous section, add a rigid joint between an axle and the hole at the front of the car.

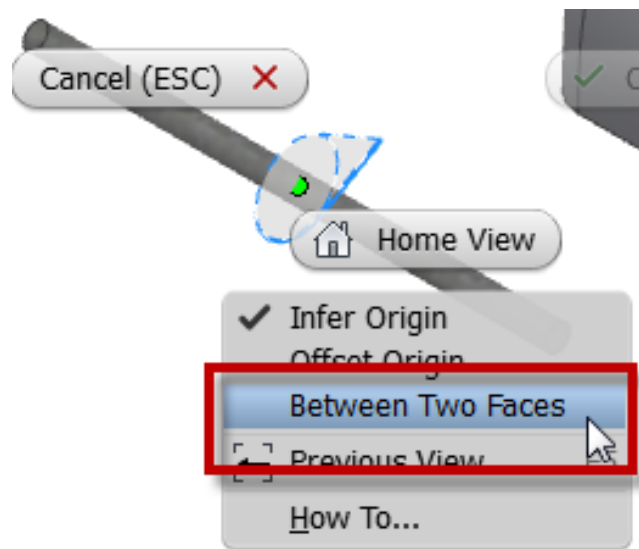


4. On the Relationships panel, click Joint.
5. Select the center of the axle.

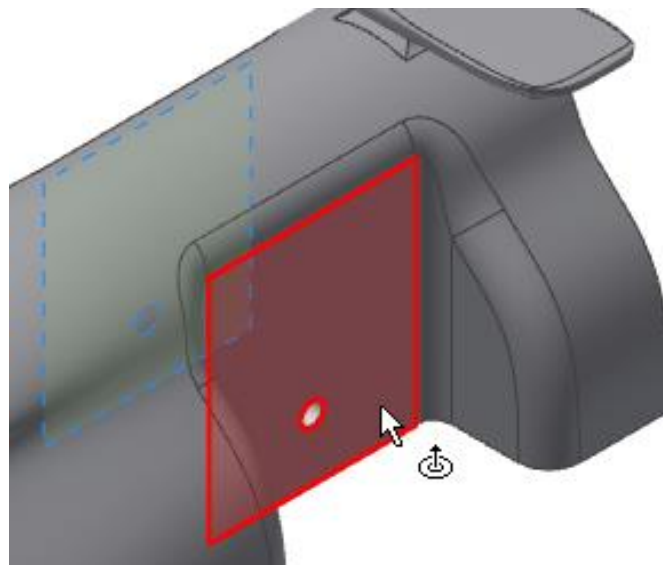




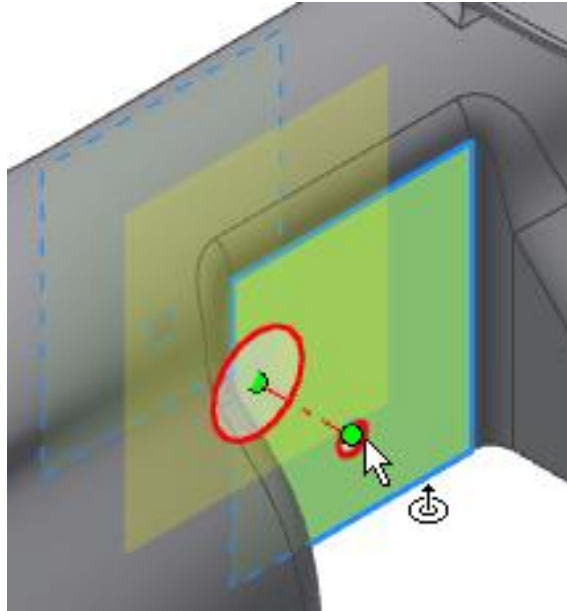
6. Right-click, then click Between Two Faces.



7. Select the front face of the block beside the hole. Then, select the opposite face.



8. For the origin, select the edge as shown.



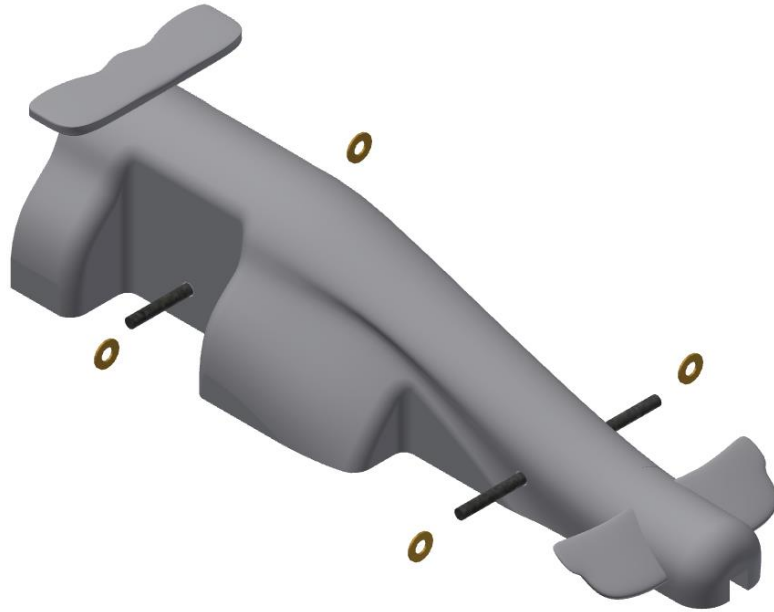
9. Click OK.

10. Return to the Home view and click Top. Review the assembly to make sure the axes are centered.

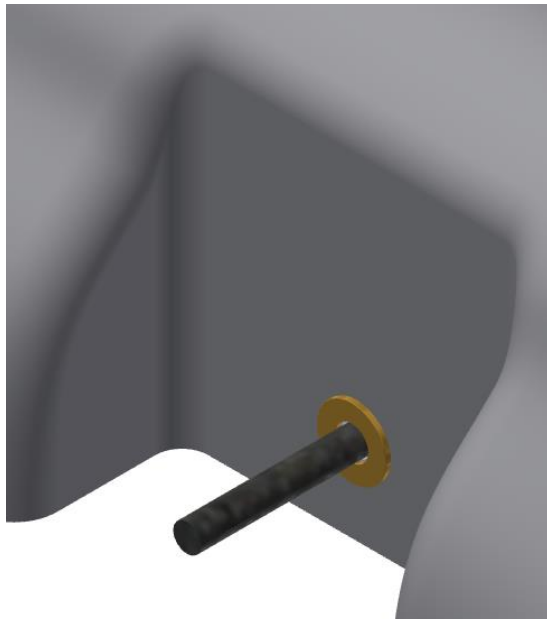


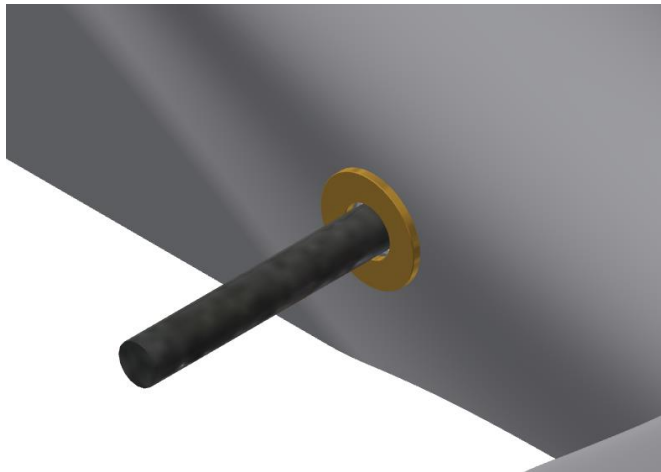
11. Return to the Home view.

12. Place four Inventor F1 in Schools Washer parts in the assembly.

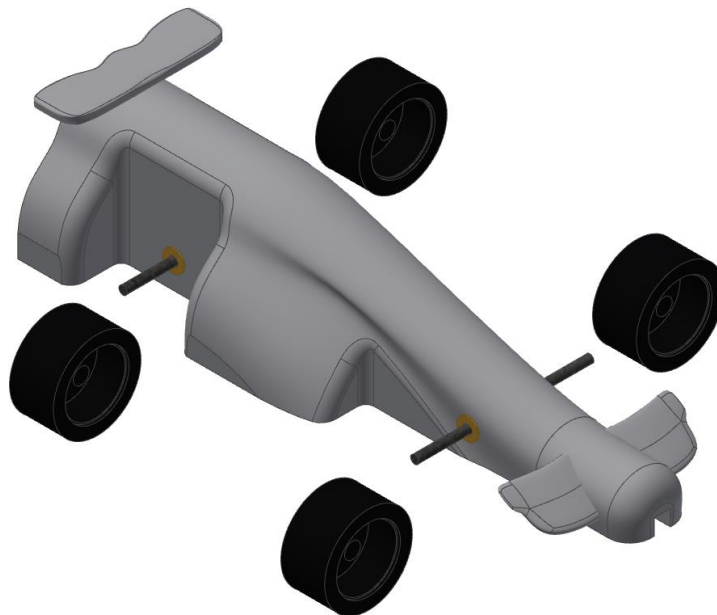


13. On the Relationship panel, click Constrain. Use the Insert type to assemble the four washers. The rear washers can be flush to the car body, but the front washers should have a **0.5mm** offset to prevent any interference with the curved surface of the car body.

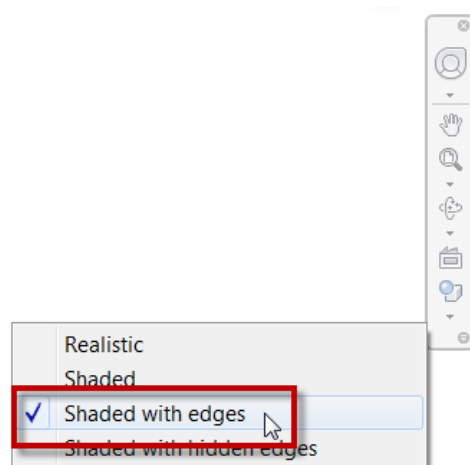




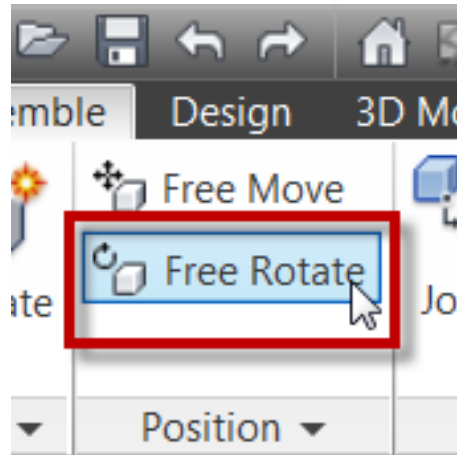
14. Place four Inventor F1 in Schools Wheel parts in the assembly.



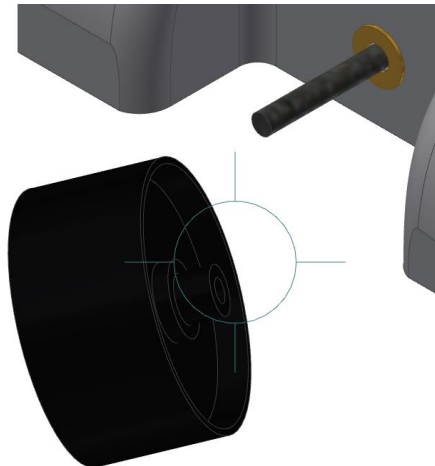
15. On the Navigation toolbar, from the list, check that Shaded with Edges is selected. This will make it easier to select the edges on the black wheels.



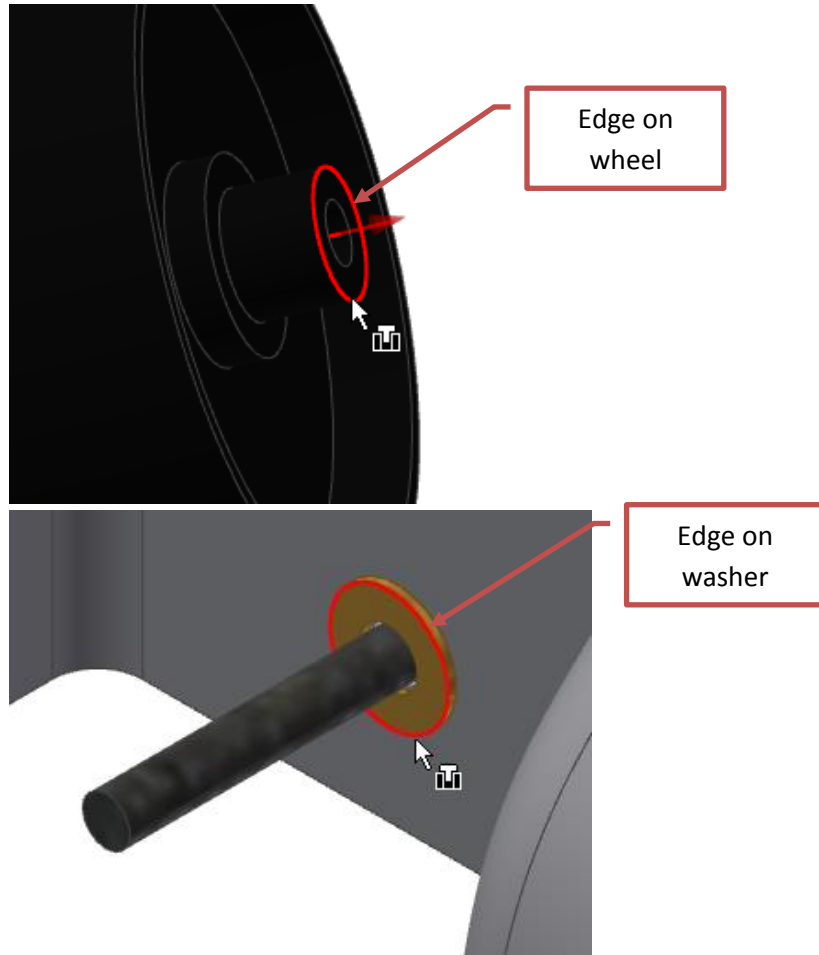
16. On the Position panel, click Free Rotate.



17. Select the rear wheel and rotate as shown. This will make it easier to select the correct edges.



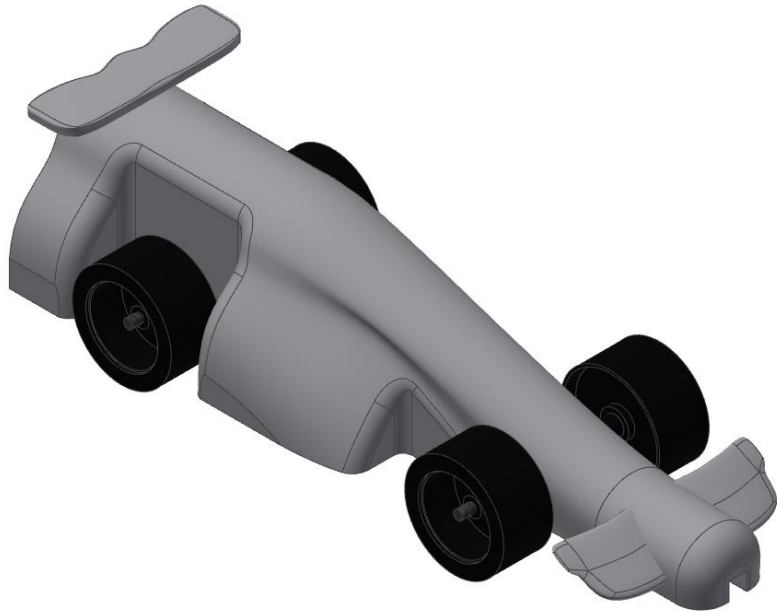
18. On the Relationship panel, click Constrain. Use the Insert type to align the wheel to the axle. Select the edge on the wheel then the edge on the washer and use an offset of **1.5mm**.



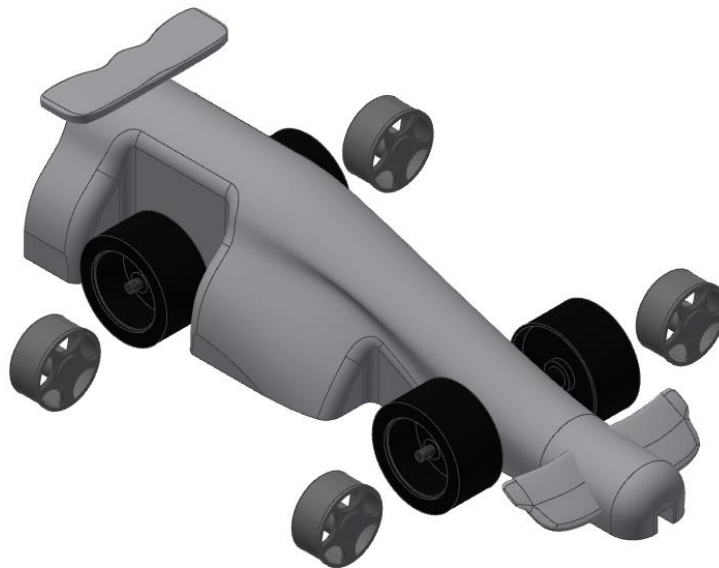
19. Click OK.



20. Repeat this workflow for the other three wheels.



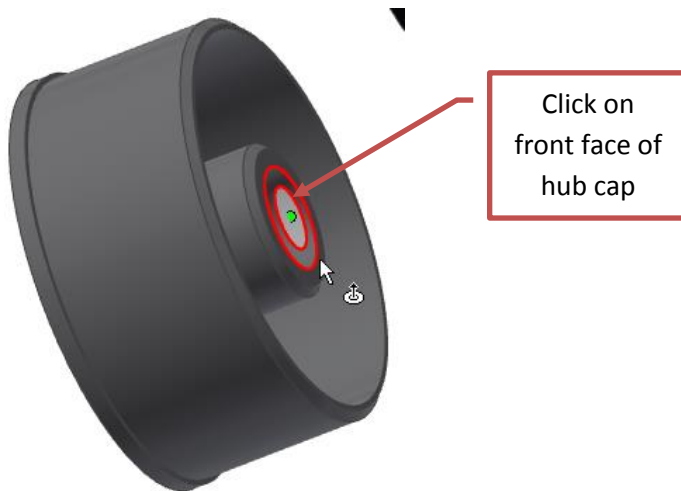
**Note:** The wheel is free to rotate around the axle. In the following steps you assemble the hubcap to the wheel using a rigid joint so that the wheel and hubcap rotate together.  
21. Place four Inventor F1 in Schools Hubcap parts in the assembly.



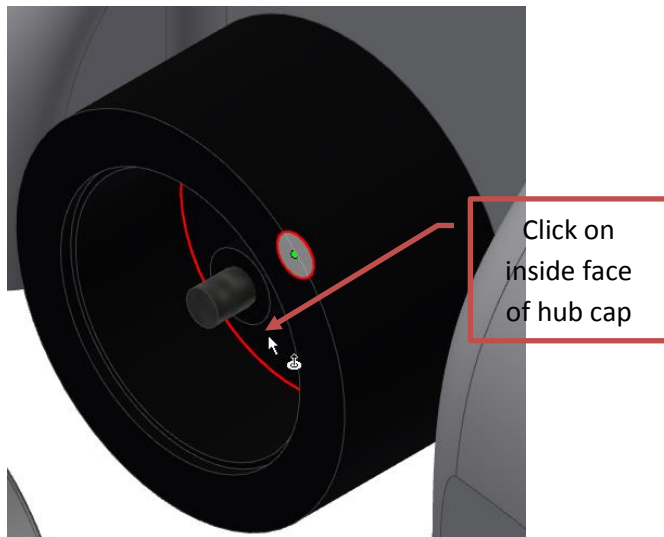
22. Use Free Rotate to align the hubcap as shown.



23. On the Relationship panel, click Joint. Use a Rigid joint type to align the hubcap origin to the wheel origin.



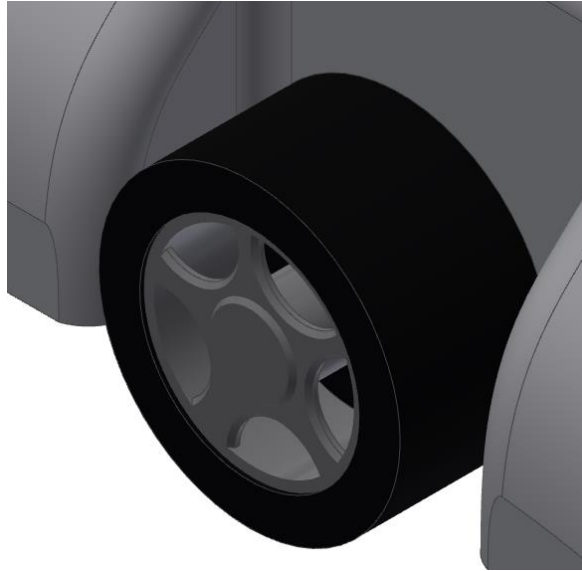
Click on front face of hub cap



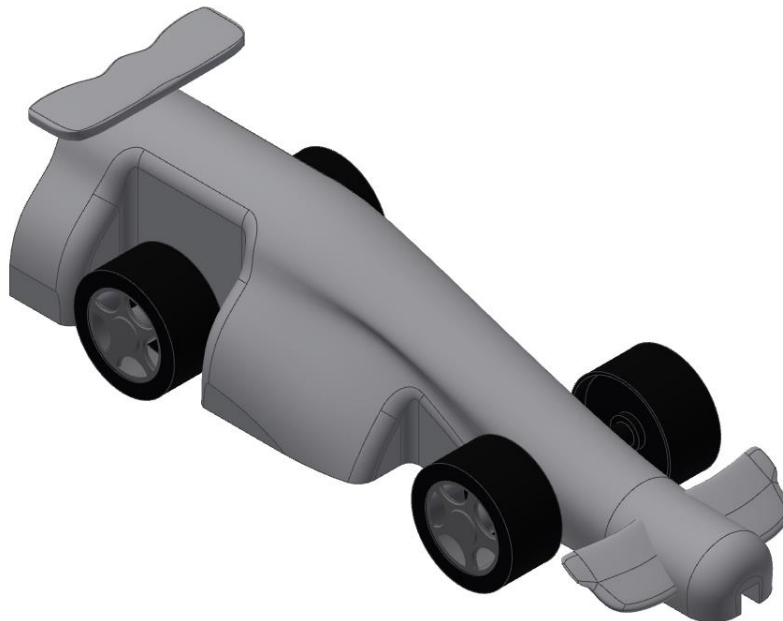
Click on inside face of hub cap



24. Click OK. Click and drag on the wheel to check that the wheel and hubcap rotate together.

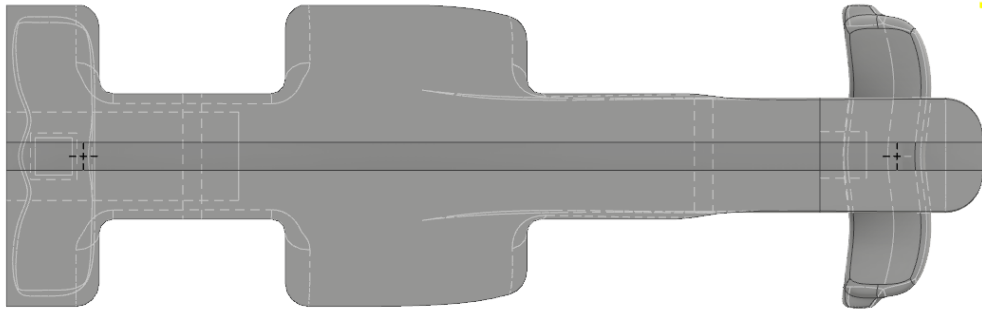


25. Repeat this workflow for the other three hub caps.



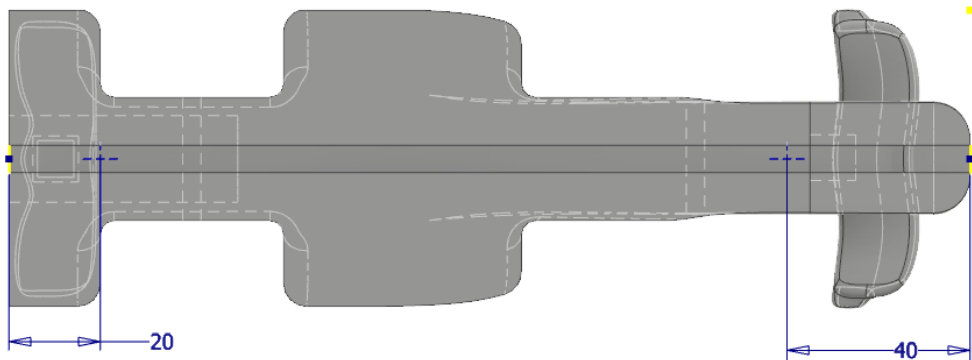
26. Right-click the car body then click Open.

27. Start a new 2D sketch on the horizontal face of the tether guide slot, then create 2 points on the sketch as shown.



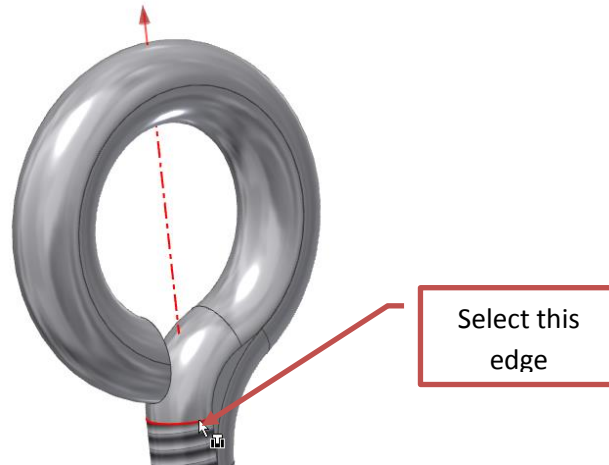
28. Constrain the points to the middle of the tether guide slot. Depending on the position of the car body, this will be a vertical or horizontal constraint.

29. Add dimensions to position the points according to the regulations—which states that the front guide must be in front of the front axle and the rear guide behind the rear axle.

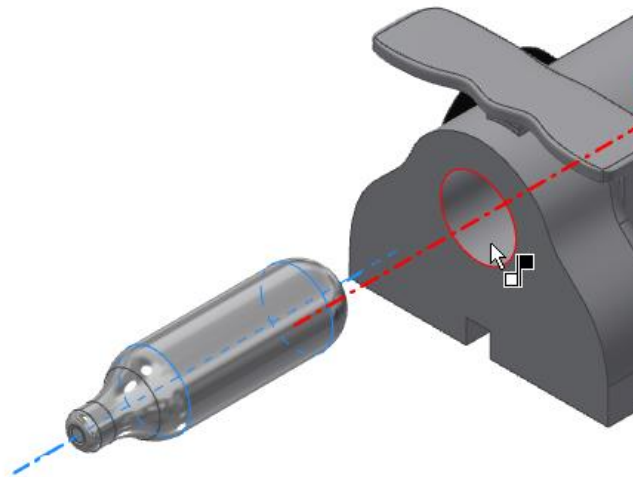


30. Press **H** to start the Hole tool, then place two holes **0.75mm** diameter and **5mm** deep.  
31. Press OK, then return to the Home view.  
32. Save the file.  
33. Return to the assembly file.

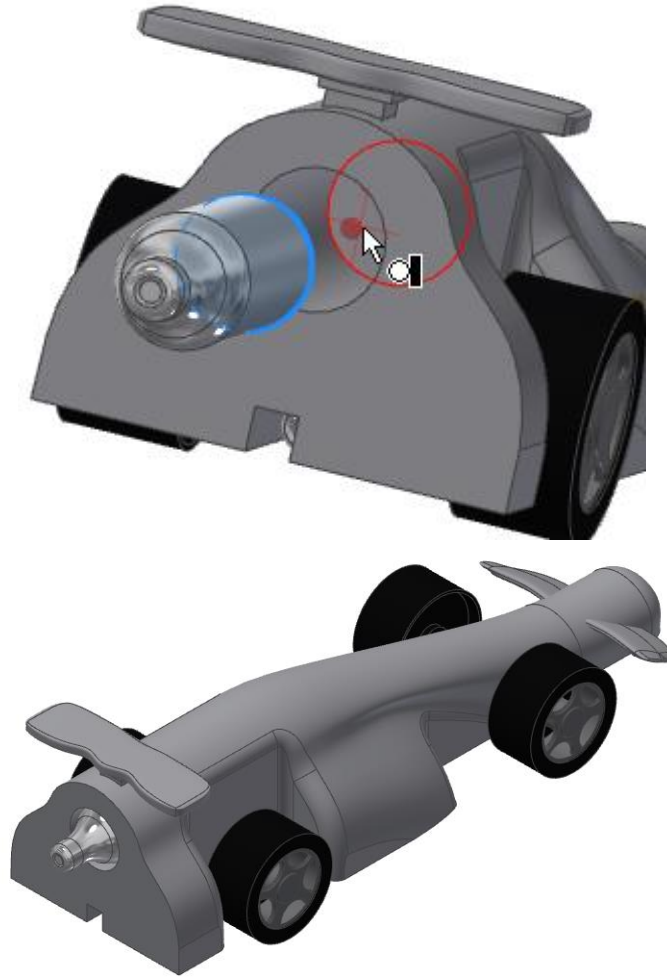
34. On the Relationship panel, click Constrain. Use the Insert type to insert the guides into the drilled holes. For the selection on the guide, use the thread edge closest to the circular end of the tether.



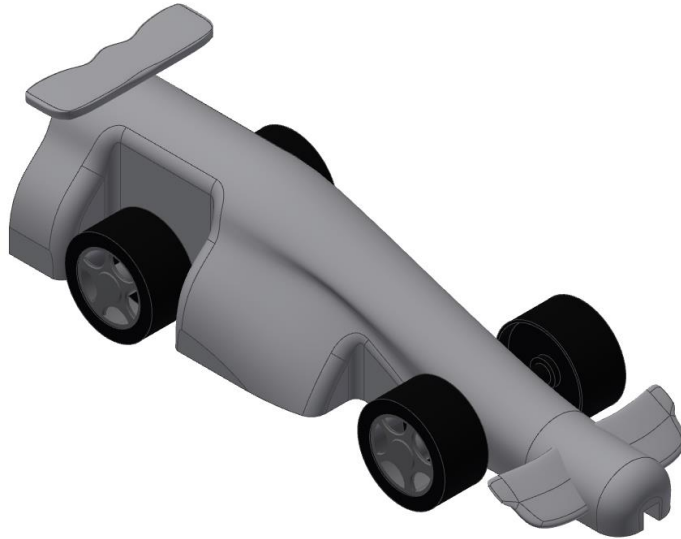
35. If required, rotate the guides to align with the slot.  
36. Place an Inventor F1 in Schools Hub CO2 Cartridge part in the assembly.  
37. On the Relationship panel, click Constrain. Use the Mate type to align the center line of the cartridge and the chamber.



38. Drag the cartridge out of the chamber, then use a tangent constraint to align the rounded end of the cartridge with the flat end of the chamber.



39. Save the file.



## Activity 7: Check for Regulation Compliance

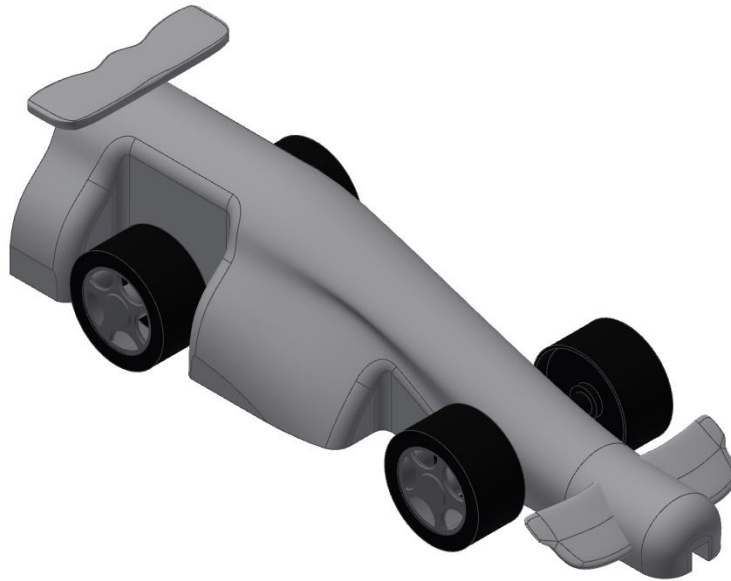
In this activity, you check if your design complies with some of the F1 in Schools Regulations. It is vital to make sure your car is legal. There are some critical Regulations that it has to comply with, and some non-critical Regulations will just cause you to lose points on if your car does not comply. For video instructional support, download the step-by-step video tutorials for this activity.

### Step 1: Review the Regulations

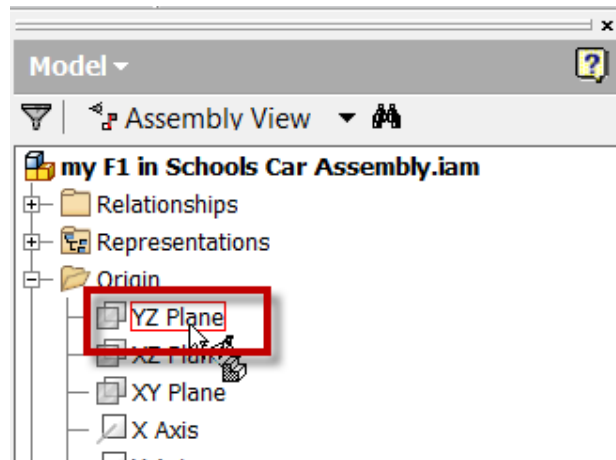
1. The current F1 in Schools Technical Regulations can be found on your in-country F1 in Schools website. For a list of in-country websites visit - <http://www.f1inschools.com/international-sites/>
2. Those highlighted in yellow are critical technical regulations and must be followed or your car will not be eligible.

## Step 2: Check the Model for Regulation Compliance

1. The assembly of your F1 in Schools car should be open.



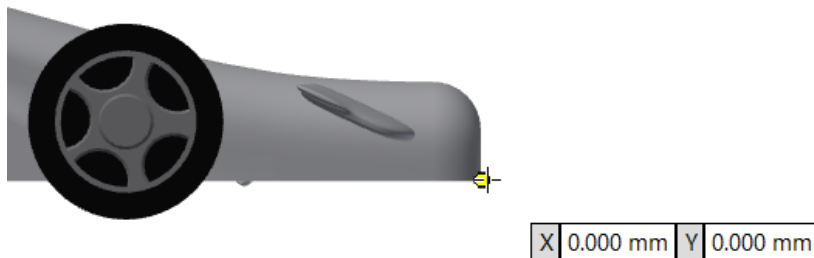
2. On the Sketch tab > Sketch panel, click Start 2D Sketch.
3. In the browser, expand the Origin folder then click YZ Plane.



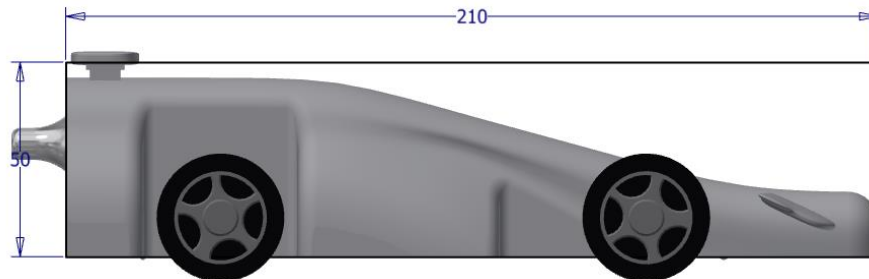
- On the ViewCube, click Left.



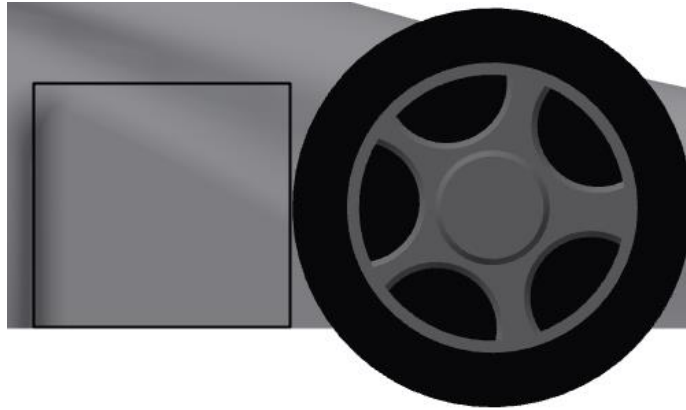
- On the Create panel, click Rectangle.
- At the bottom right corner of the car block, click to start the rectangle. This is the origin of the model.



- Drag to the left and upwards to start the rectangle.
- Enter **50**, press Tab, then enter **210**. Press Enter, then click OK.

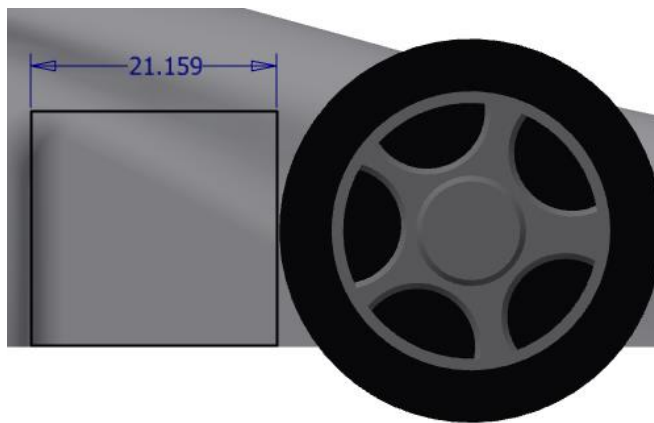


- Review the sketch and the car body only. Does it fit within the rectangle? The regulation requires that the car cannot be longer than **210mm** and this design complies with the regulation.
- On the Exit panel, click Finish Sketch.
- In the browser, right-click the sketch, then click Visibility.
- On the Sketch tab > Sketch panel, click Start 2D Sketch.
- In the browser, expand the Origin folder, then click YZ Plane.
- Sketch a rectangle behind the front wheel arch as shown.



**Note:** The Regulations state that there must be a 15mm exclusion zone behind the front wheels.

15. Add a horizontal dimension to the sketch. The dimension exceeds **15mm**, so the current design complies with the regulations.



16. On the Exit panel, click Finish Sketch.
17. Continue using this technique to check all the Regulations listed. If your car does not comply, you can double click the feature in the history to edit.

## Activity 8: Create Detailed Drawings of Your F1 Car

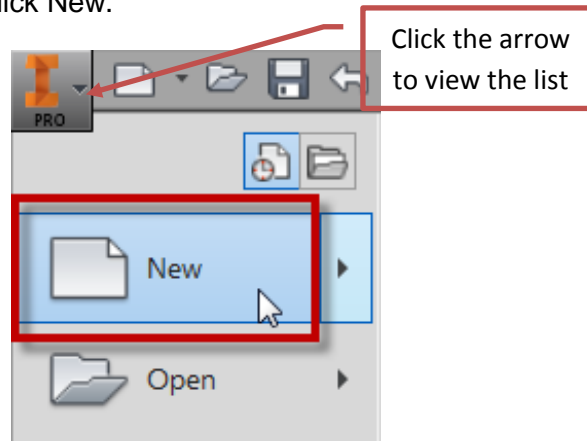
In this activity, you will create orthographic drawings and add section views, detailed dimensioning, and exploded views. For video instructional support, download the step-by-step video tutorials for this activity.

Our tutorial serves to provide you with an overview of the different Fusion 360 technical drawing tools and capabilities. F1 in Schools Competition Regulations stipulate that you should produce a 3<sup>rd</sup> Angle Projection Orthographic drawing. We recommend that you research the requirements of this specific Engineering Drawing standard before finalizing the Orthographic drawing to be included with your submission.

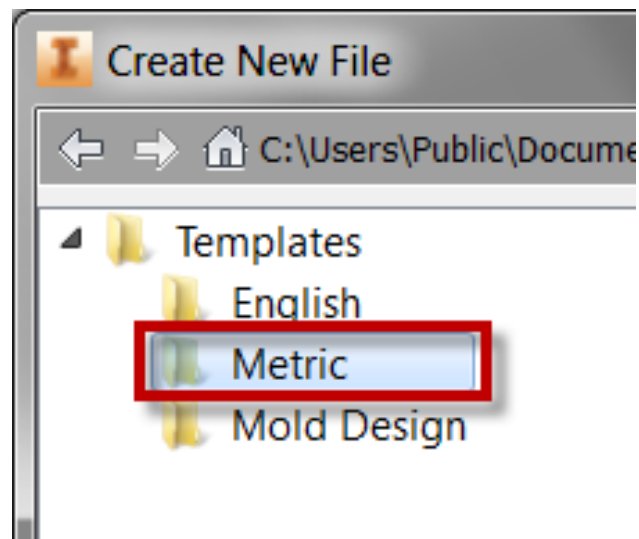


## Step 1: Create a New Drawing File

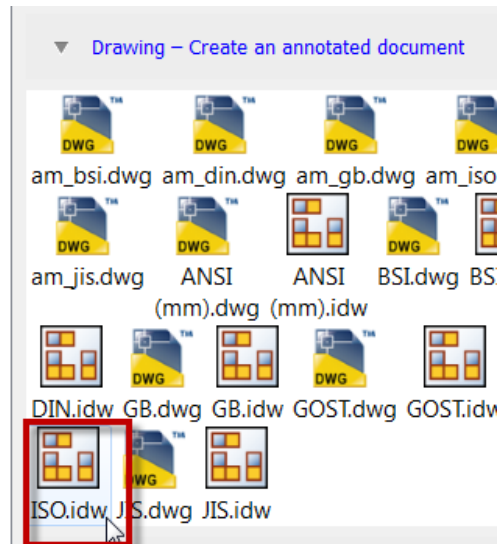
1. On the Application menu, click New.



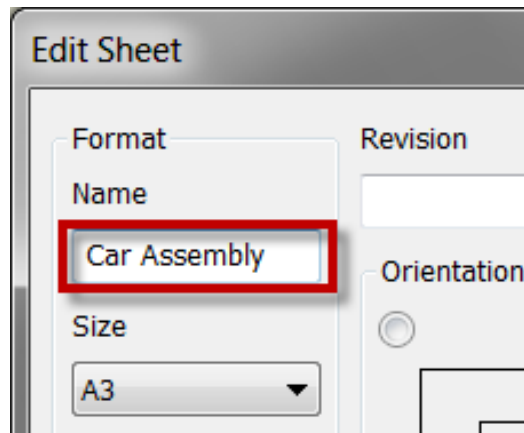
2. On the Create New File dialog box, under Templates, select Metric.



3. Under Drawing, select ISO.idw.



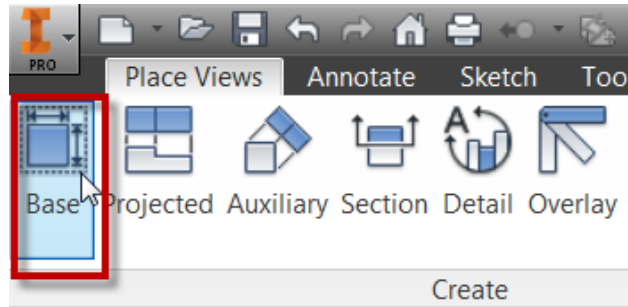
4. Click Create.
5. Save the file as **my\_F1 in Schools Car Assembly**.
6. In the browser, right-click Sheet:1 then select Edit Sheet.
7. Review the sheet size. It is an A3 (420mm x 297mm).
8. For Name, enter **Car Assembly**.



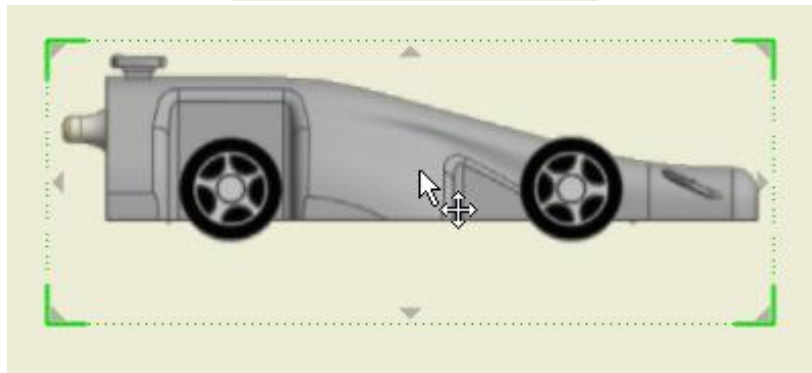
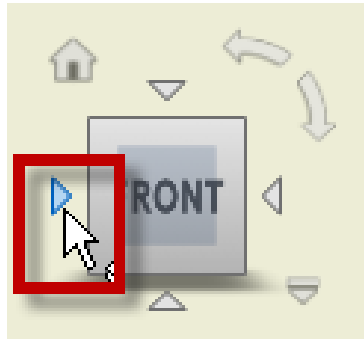
9. Click OK.

## Step 2: Create Orthographic Views

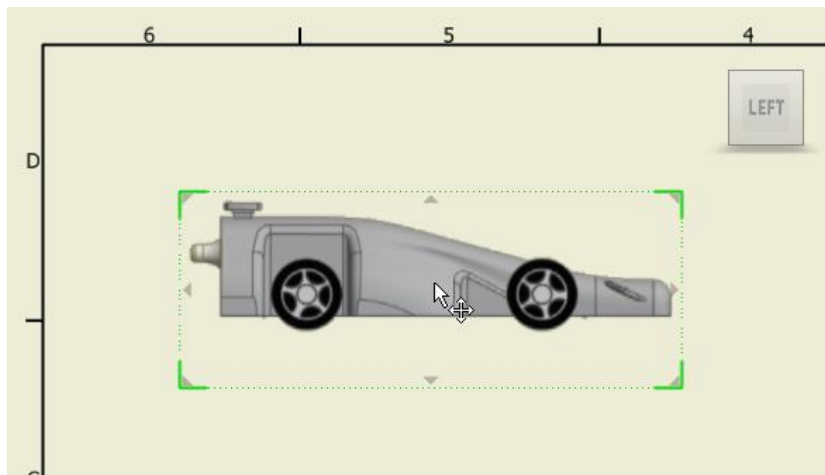
1. On the Create panel, click Base.



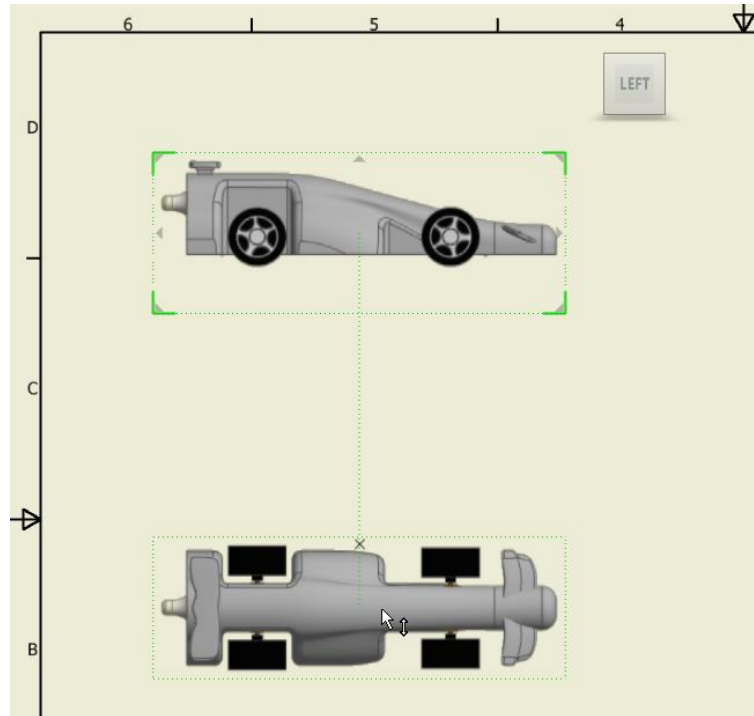
2. On the ViewCube, click the arrow on the left. The drawing preview changes.



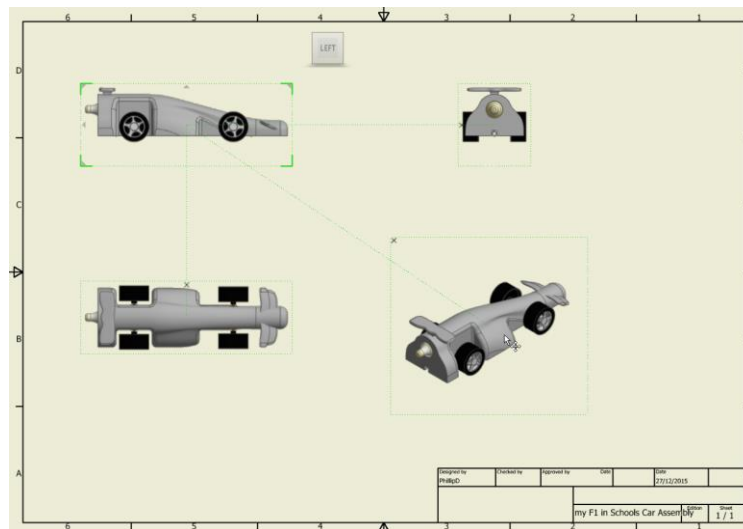
3. Drag the preview to the top left corner.



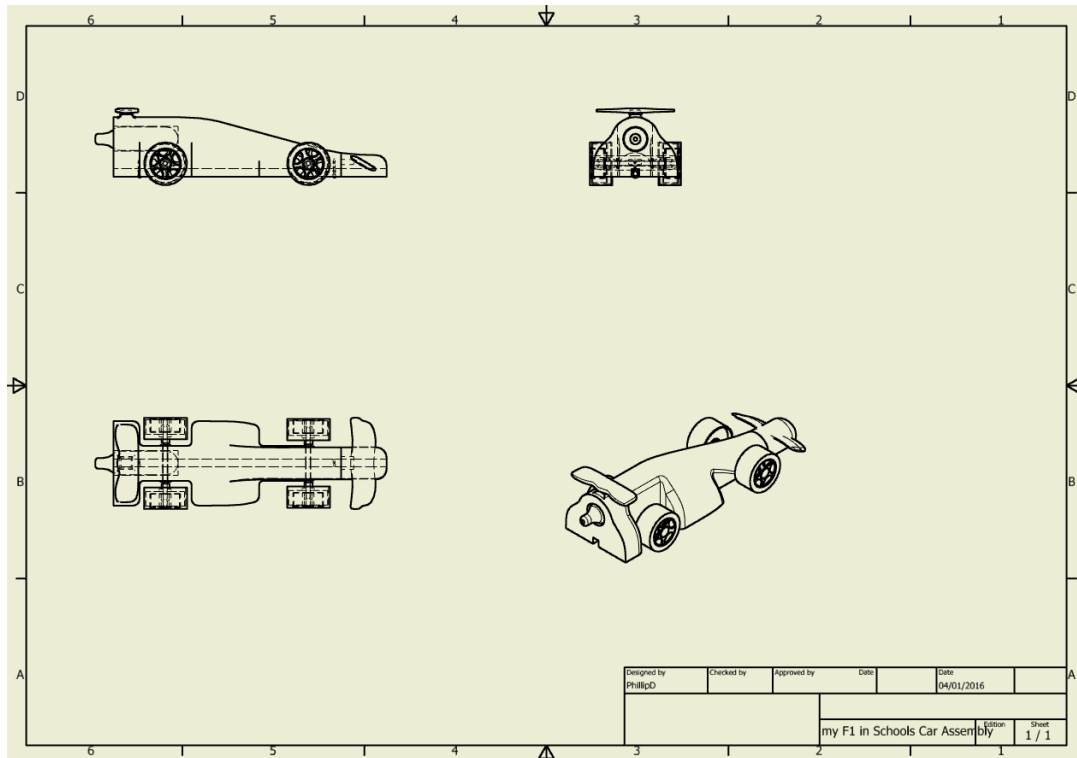
4. Move the cursor down then click to place a projected view.



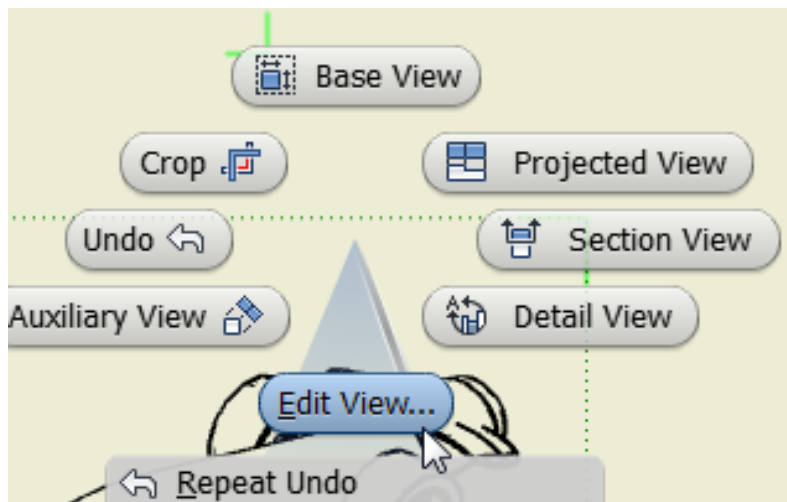
5. Repeat this process to place 2 more views as shown.



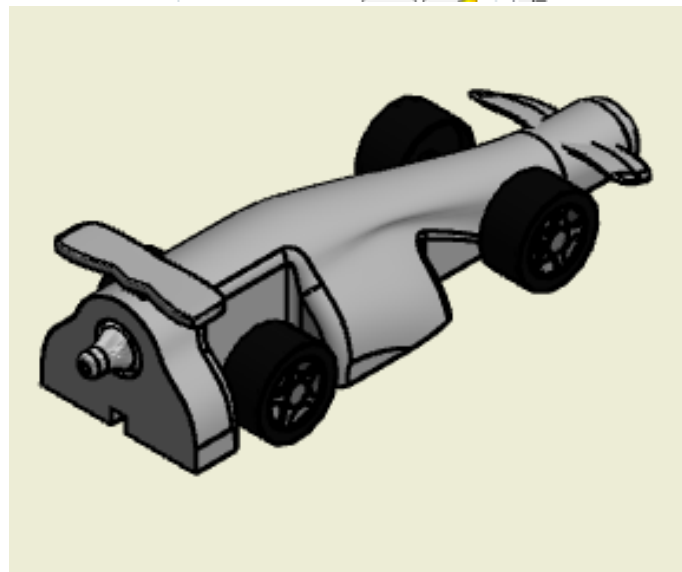
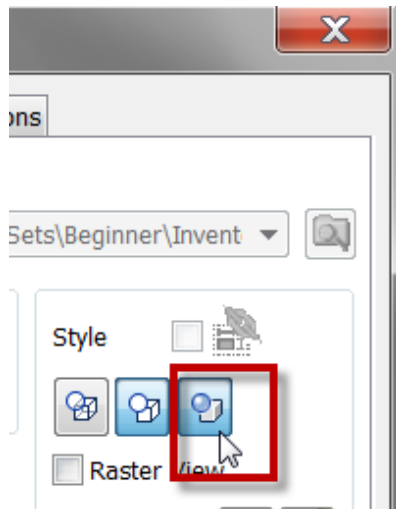
6. In the Drawing View dialog box, click OK. 4 views of the car body are created at a scale of 1:2.



7. Right-click the pictorial view then click Edit View.

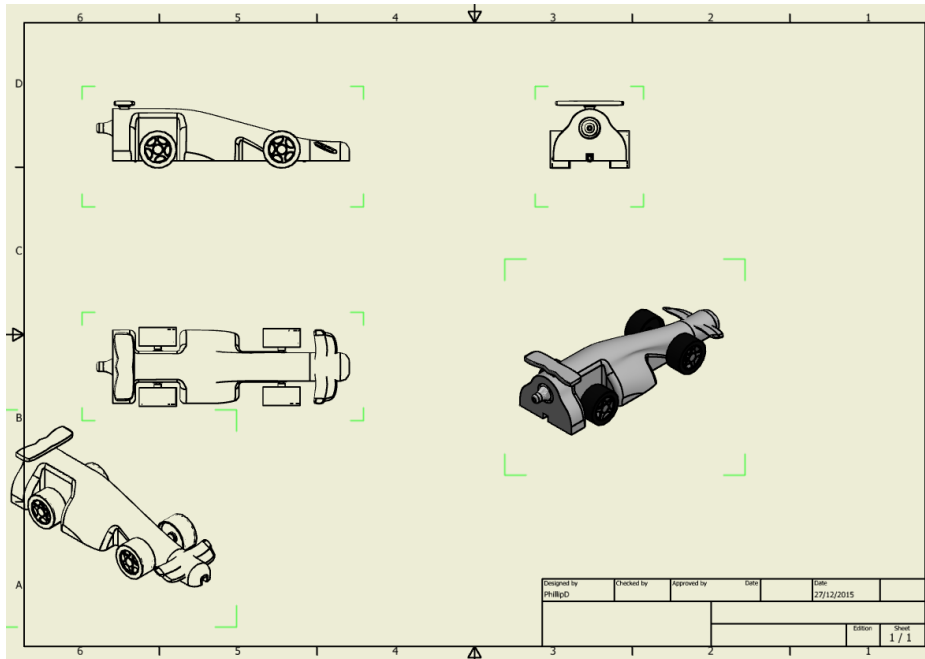


8. In the Drawing View dialog box, under Style, click Shaded then click OK.



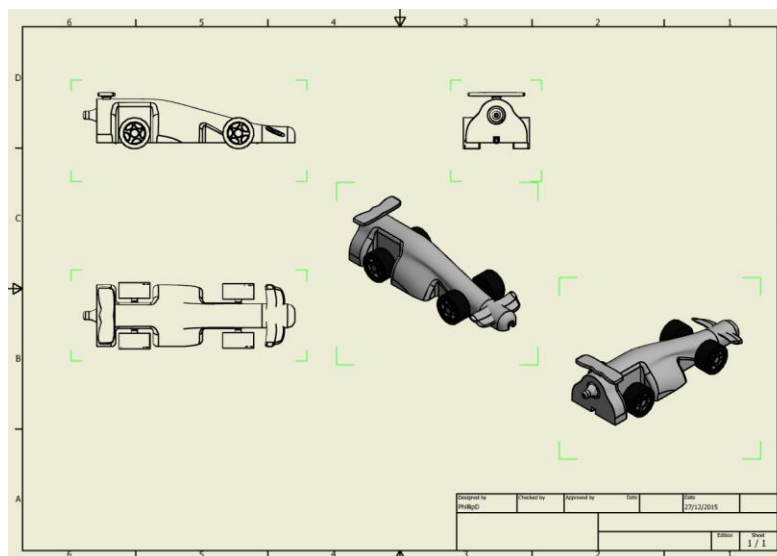
10. On the Create panel, click Projected.

11. Click and drag the front view to create another pictorial view as shown.



12. Drag the pictorial views to arrange them on the sheet as shown. Then, change the style of the new pictorial to shaded.

Note – what is shown in this tutorial is purely representative of what can be achieved. You will need to research the ISO drawing layout standard for 3<sup>rd</sup> Angle Orthographic Technical Drawings so that you can ensure your drawings comply with this standard

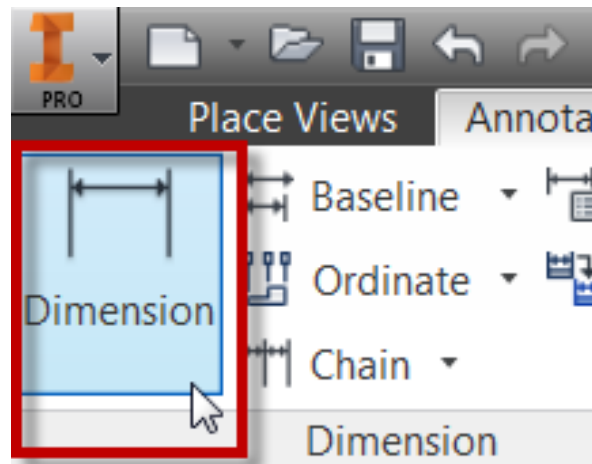


13. Save the file.

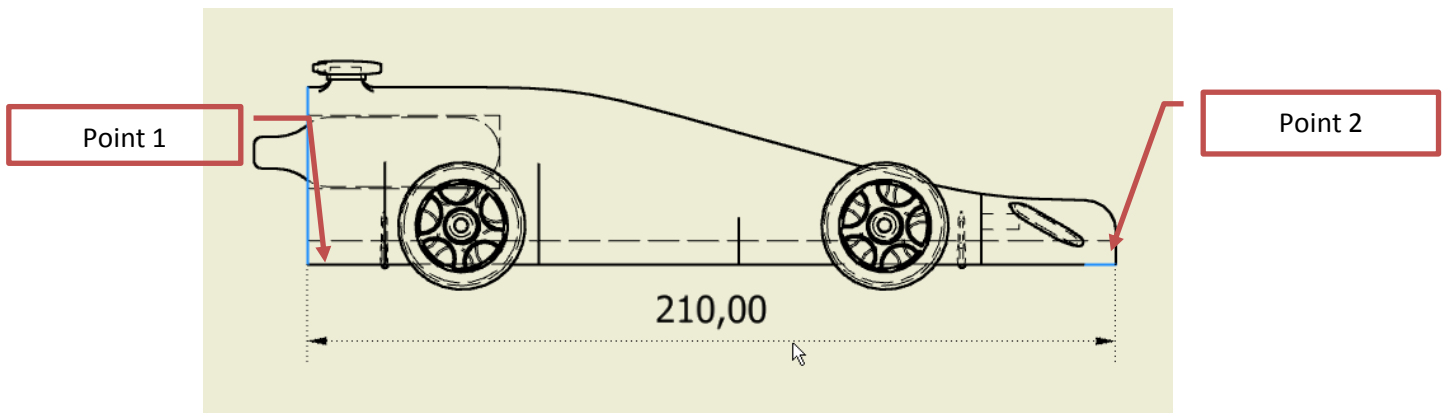
### Step 3: Adding Dimensions

As mentioned earlier, you should research the relevant drawing standards to learn more about how dimensions and other text should be properly placed on your drawing relative to the views and parts.

1. On the Annotate tab > Dimension panel, click Dimension.

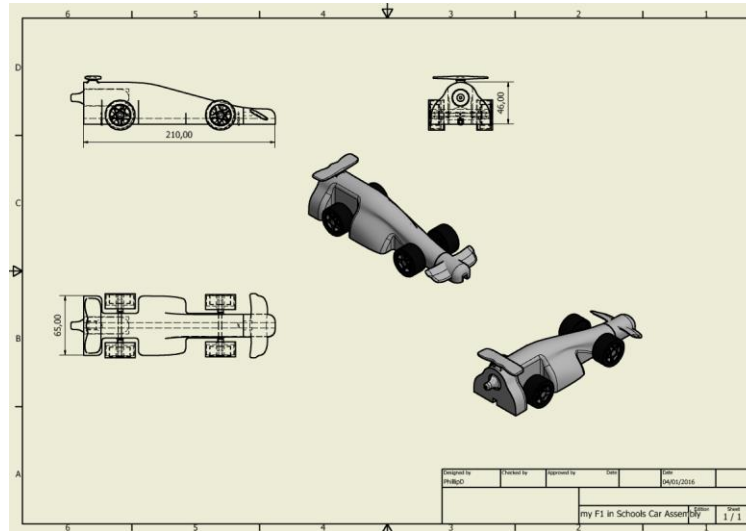


2. Click point 1 and 2 as shown, then drag the dimension away from the view until a dotted line is displayed. Click to place the dimension.



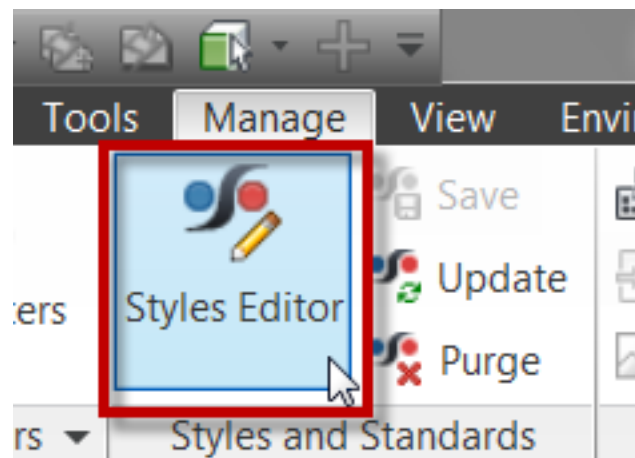


- Repeat this process for 2 more dimensions. When the dimensions are placed, right click then click OK.

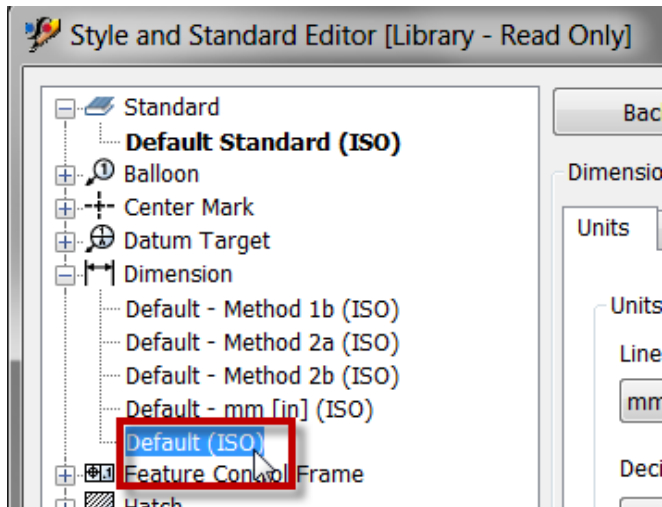


**Note:** The dimensions are displayed with a precision of 2 decimal places. You will change the dimension style to display zero decimal places.

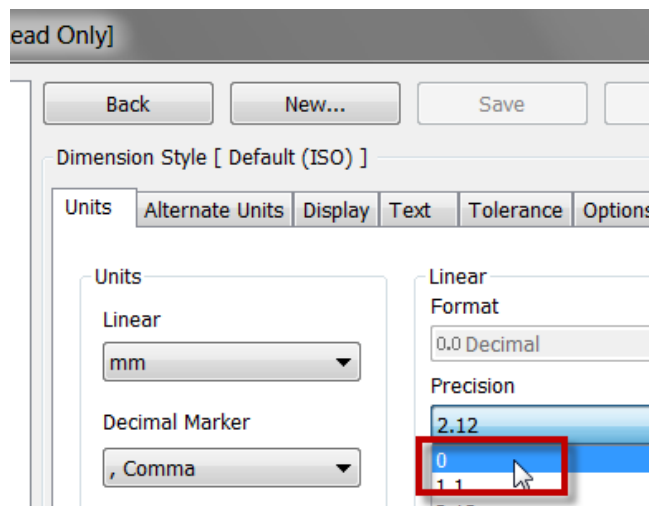
- On the Manage tab > Styles and Standards tab, click Styles Editor.



- In the left panel, expand Dimension, then click Default (ISO).



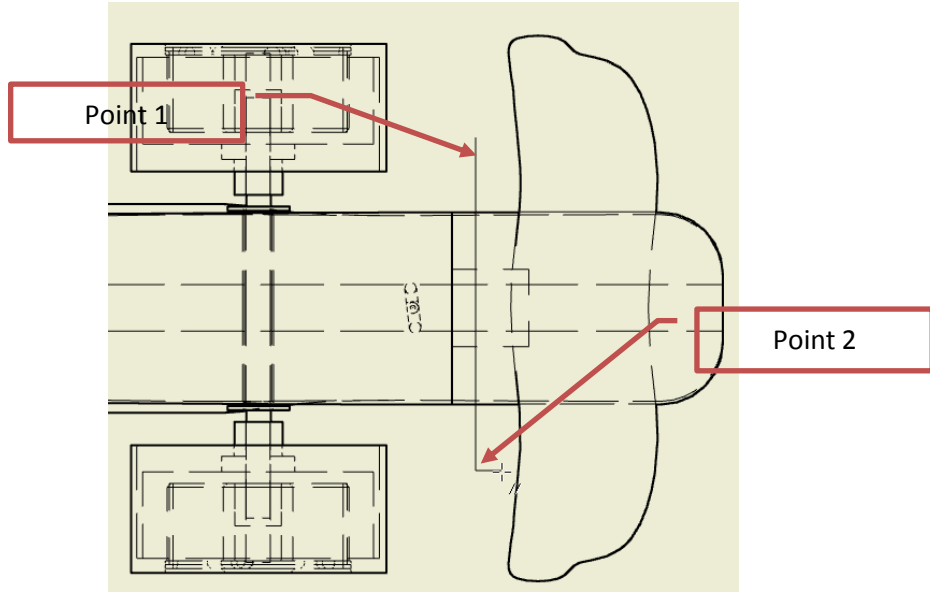
6. On the Units tab, under Linear, select 0 from the Precision list.



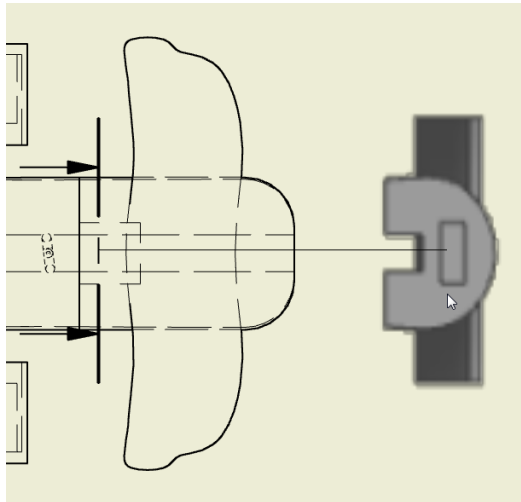
7. Click Save then click Done. The dimensions are updated to the new style setting.

#### Step 4: Create Section Views

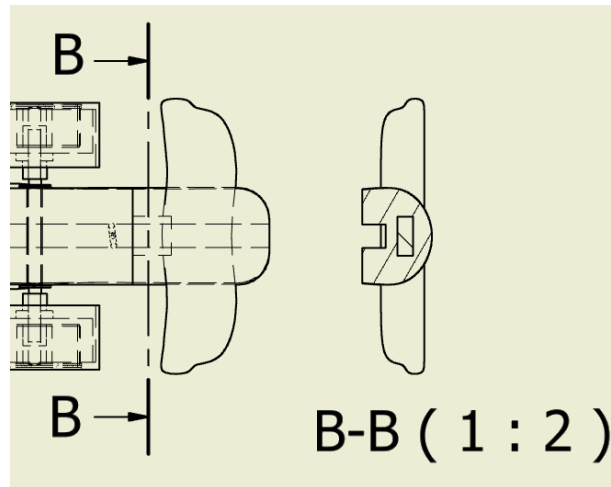
1. On the Place View tab > Create panel, click Section.
2. Click on the plan view, then click 2 points to the left of the nose cone as shown.



3. Drag the cursor to the right, and then right-click and click Continue. A preview of the view is displayed.



4. Right-click, then click OK. The section line and text can be dragged to new locations to improve the presentation of the drawing.

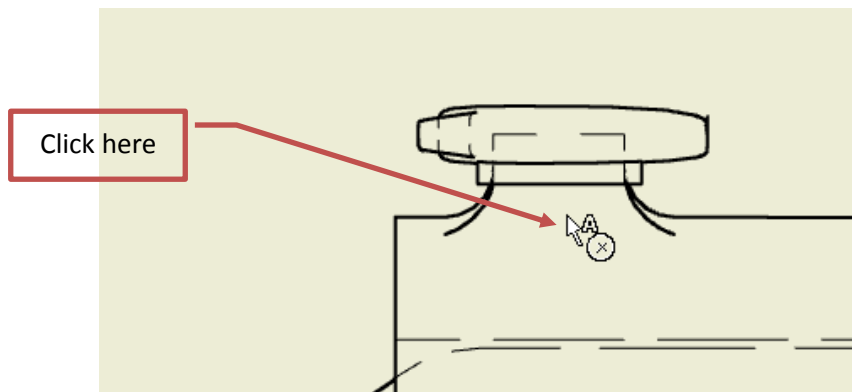


5. Save the file.

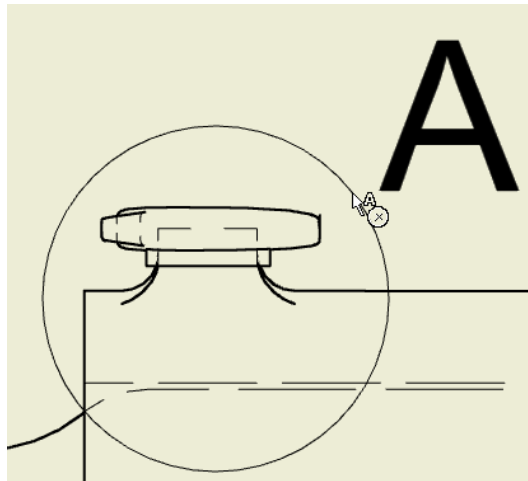
### Step 5: Create Detail Views

Detail views are created to show larger scale views of parts that are difficult to see or dimension.

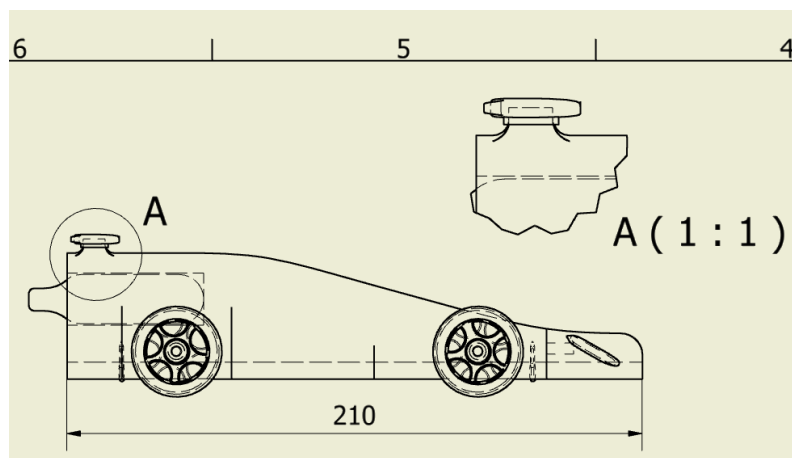
1. On the Create panel, click Detail.
2. In the front view, click below the rear wing as shown.



3. Drag the cursor to create a circle as shown.



4. Click to create the view, then drag it above the front view as shown. Move the text below the view.



5. Save the file.

## Next Steps

Now that you have successfully designed an F1 in Schools car, you can move on to the advanced tutorial. This tutorial will give you tips on how to create your own car design using different 3D modeling techniques, how to model your own wheels, and how to analyze the mass of your model.

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