

# Introduction to Autodesk Inventor for F1 in Schools

## F1 in Schools Race Car

In this course you will be introduced to Autodesk Inventor, which is the centerpiece of Autodesk's digital prototyping strategy that allows you to create and simulate a design before it is made. Autodesk Inventor is a 3D parametric modeler that allows you to easily create and modify a design. In this fun and exciting hands-on course, you will learn the basic design workflow by modeling and documenting an F1 in Schools race car.



## Course Objectives

After completing this course, you will be able to:

- Model an F1 in Schools race car.
- Create drawing views of an F1 in Schools race car.
- Assemble an F1 in Schools race car.
- Enhance the visual appeal of the car by changing the visual style of the car, adding a decal, shadows and reflections, and changing the background and adjusting the light sources.

## Student Notice

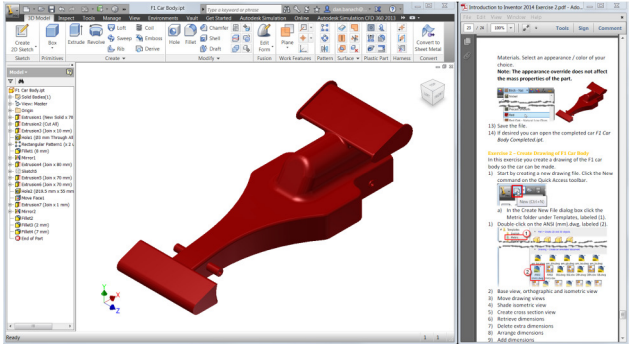
**The design and dimensions shown in this course are for demonstration purpose only. Check the F1 in Schools technical rules and regulations for design and dimension specifications.**

## Free Autodesk Software

Autodesk is pleased to support your design efforts by making our industry leading 3D design software products available to students for free. Register and then download software from <http://students.autodesk.com>

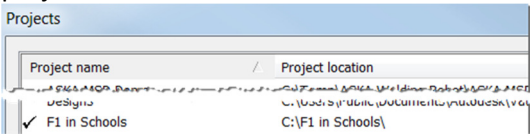
## Screen Setup

To make it easier to go through this tutorial position Autodesk Inventor and this PDF as shown on one screen and scroll through the PDF as you progress.

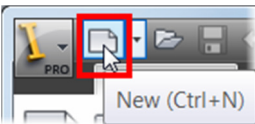


## Exercise 1: Section 1 – Getting Started

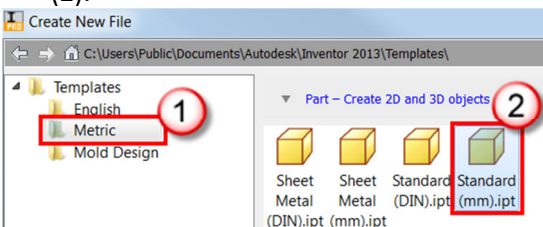
- 1) First you set where Inventor will look for files and where files will be saved. Set the active project by clicking the Projects command either on the Welcome screen or on the Get Started tab > Launch panel.
  - a) Click the Browse button at the bottom of the Project dialog box.
  - b) Navigate to the location where the exercise file were saved (the default location is *C:\F1 in Schools*) and double click on *F1 in Schools.ipj*.
  - c) In the Projects dialog box, a check mark should be to the left of the F1 in Schools project name, verifying that this is the active project file.



- 2) The first step is to create a new part file, click the New command on the Welcome dialog box or on the Quick Access toolbar.



- a) In the Create New File dialog box click the Metric folder under Templates, labeled (1).
- b) Double-click on Standard (mm).ipt, labeled (2).

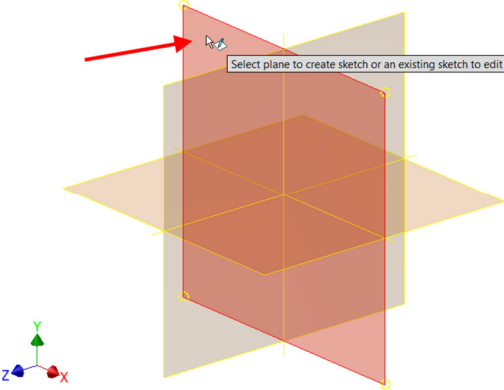


- 3) First you define where you will sketch the profile of the car. Create a new sketch by right-clicking on a blank area in the graphics window and click

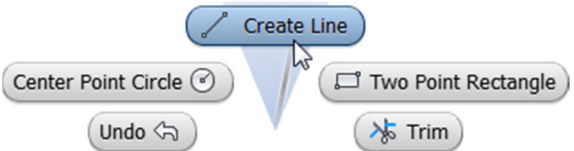
New Sketch on the marking menu.



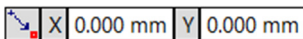
4) In the graphics window select the XY origin plane.



5) Sketch profile (outline) of the side view of the car body by right-clicking on a blank area in the graphics window and click Create Line on the marking menu, or press the L key or click Sketch tab > Draw panel > Line.



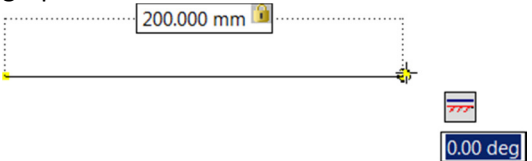
a) Click on the Origin point (0,0) in the middle of the screen, a green circle will appear when on the origin point.



b) Enter a value of 200 mm in distance value cell.

c) Press the TAB key to lock in this value.

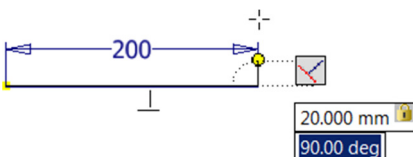
d) Move the cursor to the right and when the horizontal symbol appears, click in the graphics window.



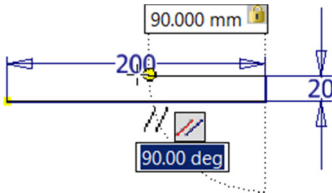
e) Enter a value of **20 mm** in distance value cell.

f) Press the TAB key to lock in this value.

g) Move the cursor to up and when the perpendicular symbol appears, click in the graphics window.



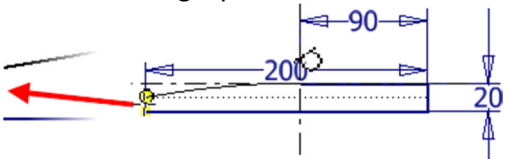
- h) Enter a value of **90 mm** in the distance value cell.
- i) Press the TAB key to lock in this value.
- j) Move the cursor to the left and when the parallel symbol appears, click in the graphics window.



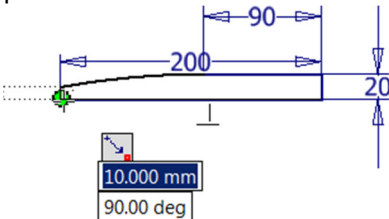
- k) Next you sketch an arc while in the line command. While still in line command, click on the left endpoint of a top horizontal line and a small gray circle will appear.



- l) Click on the small circle, and with the left mouse button pressed down, move the cursor down and to the left until it is directly above the origin point and click. When done, there should be a gap between the endpoint of the arc and the origin point.



- m) Close the profile by clicking on the origin point.



- n) Finish the Line command by right-clicking on a blank area in the graphics window and click OK on the marking menu.

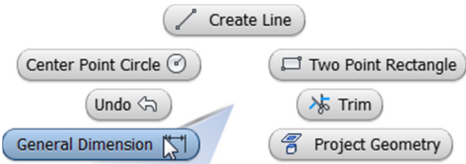


- 6) When sketching, you want to fully constrain sketches by adding constraints and dimensions. Notice on the lower corner of Inventor's Status Bar Inventor tells you how many dimensions or constraints are required to fully constrain the sketch.

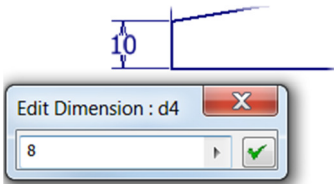
1 dimensions needed

- 7) Next you add the missing dimension. Start the dimension command by right-clicking on a blank area in the graphics window and click General Dimension on the marking menu, or press the D

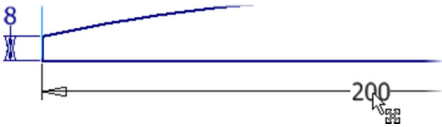
key or click Sketch tab > Constrain panel > Dimension.



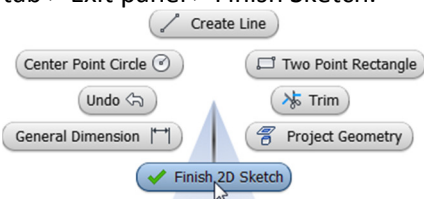
- a) In the graphics window, select the left vertical line, and then move the cursor to the left and then click to place the dimension.
- b) Change the length of the line by entering **8** in the Edit Dimension dialog box and then press ENTER or click the green check mark in the dialog box. Since no unit was entered, the default unit of the document will be used, in this case is millimeters.



- 8) Next you move the 200 mm dimension, as it is going through the vertical lines.
  - a) Press the Esc key to exit the Dimension command.
  - b) Move the cursor over the 200 mm dimension until the four arrows appear, and then click and drag the dimension below the bottom horizontal line.



- 9) The sketch dimensions are parametric, meaning that the value of the dimension controls the size of the geometry. Change the size of the 200 mm by double-clicking on it and enter **250** in the Edit Dimension dialog box and then press Enter or click the green check mark in the dialog box.
- 10) Change the 250 mm dimension back to 200 mm by double-clicking on the 250 mm dimension and enter **200** in the Edit Dimension dialog box and then press Enter.
- 11) Exit the sketch environment by right-clicking on a blank area in the graphics window and click Finish 2D Sketch on the marking menu or click Sketch tab > Exit panel > Finish Sketch.



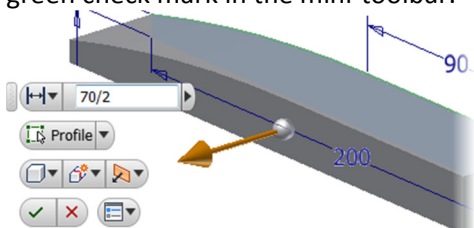
Finish the

- 12) If needed, scroll the wheel on the mouse to zoom in and out. Hold down the wheel on the mouse to pan the screen.

- 13) Next you extrude the profile to create one half of the car body. Start the extrude command by clicking on any geometry in the sketch and click Create Extrude from the mini-toolbar, or press the E key or click 3D Model tab > Create panel > Extrude.



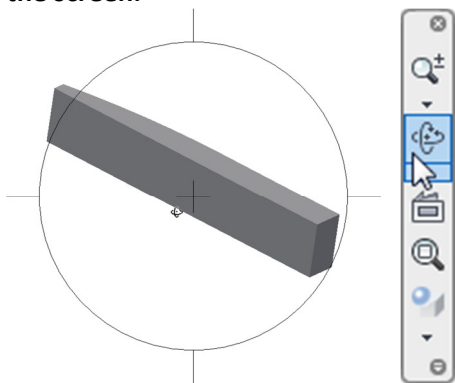
- a) Next you extrude the profile one half of its overall width, which will be 70 mm. In the value area in the mini-toolbar you can enter **35** or **70/2** and then press ENTER or click the green check mark in the mini-toolbar.



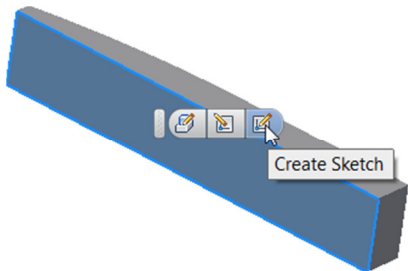
- 14) Next you remove material to refine the shape the car body.

- a) Rotate the viewpoint so you can see the bottom of the car by pressing down the F4 key and click and drag inside the circle that appears in the middle of the screen and then release the F4 key.

**TIP: To change the viewpoint you can also use the Free Orbit command on the Navigation bar, which is on the right side of the screen.**

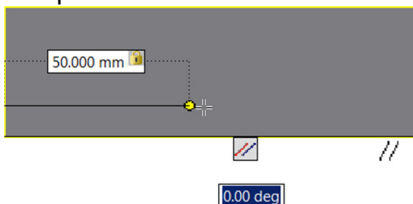


- b) Create new sketch by clicking on the bottom plane and click the Create Sketch on the mini-toolbar.

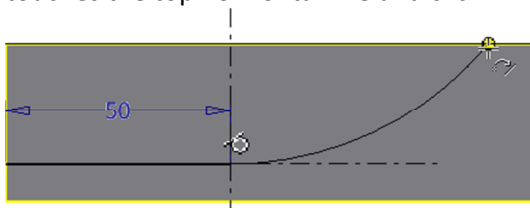


- 15) Sketch for the area that will be removed, right-click on a blank area in the graphics window and

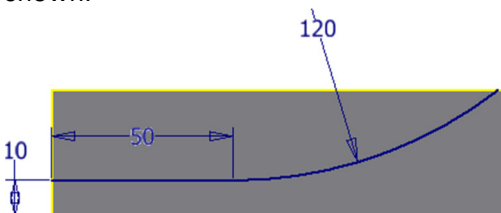
click Create Line on the marking menu and select the left-vertical line about a quarter of the way up and place a **50 mm** line.



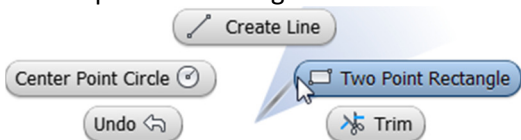
- a) While still in the line command, click on the right endpoint of the line and a small gray circle will appear. Click on the small circle, and with the left mouse button pressed down, move the cursor up and to the right until it touches the top horizontal line and click.



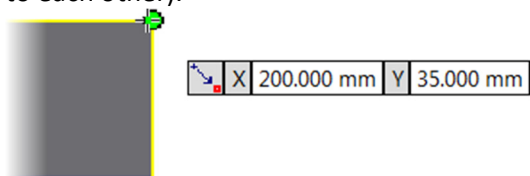
- b) Start the dimension command and add a **10 mm** linear and **120 mm** radius dimension as shown.



- 16) Next draw a rectangle by right-clicking on a blank area in the graphics window and click Two Point Rectangle on the marking menu, or click Sketch tab > Draw panel > Rectangle.

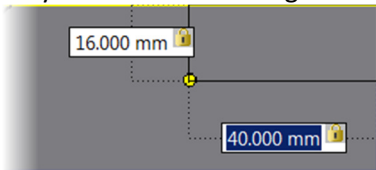


- a) Place the first point of the rectangle by clicking on the upper-right corner of the sketch. When the cursor is over the corner, a green circle appears, indicating a coincident constraint will be applied (the points are tied to each other).

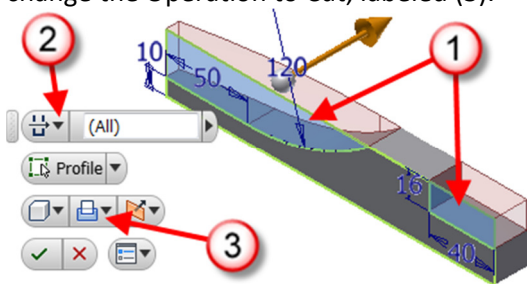


- b) Move the cursor down and to the left and enter **40** and press the Tab key to lock in the dimension.
- c) Enter **16** and press the Tab key to lock in the dimension and then press Enter to create the

fully constrained rectangle.

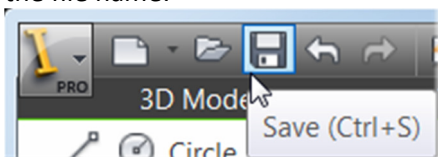


- d) Exit the sketch environment by right-clicking on a blank area in the graphics window and click Finish 2D Sketch on the marking menu or click Sketch tab > Exit panel > Finish Sketch.
- e) Next you extrude the profile to remove material from the car body. Start the extrude command by clicking on any geometry in the sketch and click Create Extrude from the mini-toolbar, or press the E key or click 3D Model tab > Create panel > Extrude.
- f) Since there are multiple closed profiles select in the top two areas that you sketched to remove material from, labeled (1).
- g) In the mini-toolbar or Extrude dialog box change the Extents to All, labeled (2).
- h) In the mini-toolbar or Extrude dialog box change the Operation to Cut, labeled (3).



- i) Complete the command by clicking the green check mark in the mini-toolbar or click OK in the Extrude dialog box.

- 17) Save the file by clicking the Save command on the Quick Access toolbar and enter *F1 Car Body* for the file name.



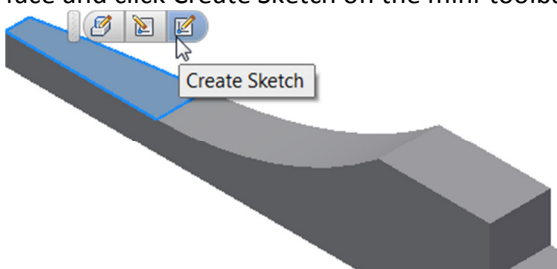
### Exercise 1: Section 2 – Define Car Body's Shape

In this step you create a wheel extension that will extend the wheel away from the body.

Continue working on the same file or if you were not able to complete the previous section, open the file *Ex1 Sec2 F1 Car Body.ipt*.



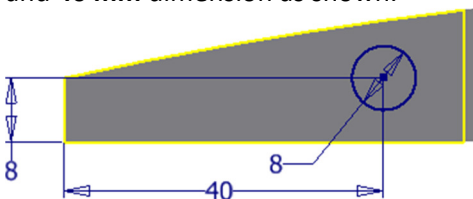
- 1) Create new sketch by clicking on the back planar face and click Create Sketch on the mini-toolbar.



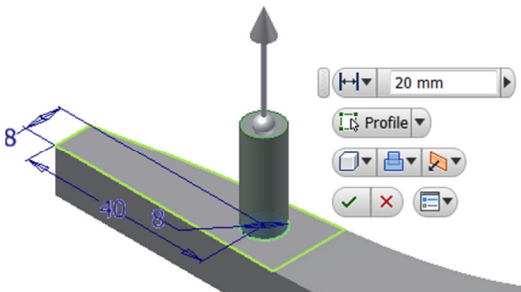
- 2) Next draw a circle by right-clicking on a blank area in the graphics window and click Center Point Circle on the marking menu, or press the C key or click Sketch tab > Draw panel > Circle.



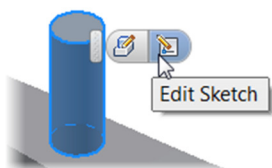
- a) Click a point on the right side of the planar face and enter **8** and to create the circle press the Enter key.
- 3) Start the dimension command and add an **8 mm** and **40 mm** dimension as shown.



- 4) Finish the sketch and extrude the circle **20 mm**, click inside the circle to define the profile to extrude.

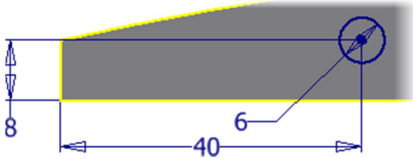


- 5) If needed you can edit the sketch by selecting on the face of the feature. In this step you edit the sketch of the extruded circle. Select the circular face of the extruded circle and click Edit Sketch on the mini-toolbar.

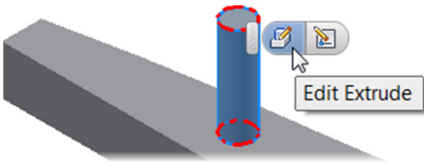


- a) Double-click on the 8 mm diameter dimension and change its value to **6 mm**.
- b) Click the green check mark in the Edit Dimension dialog box and the sketch should

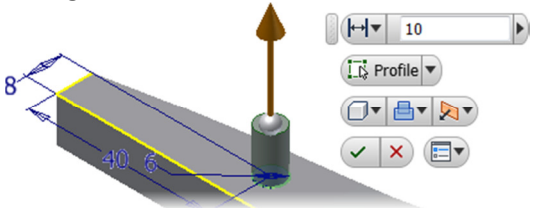
resemble the following image.



- c) Exit the sketch environment by right-clicking on a blank area in the graphics window and click Finish 2D Sketch on the marking menu or click Finish Sketch on the Sketch tab > Exit panel.
- 6) You can also edit a feature selecting on the face of the feature. In this step you edit the extruded distance of the circle. Select the circular face of the extruded circle and click Edit Extrude on the mini-toolbar.



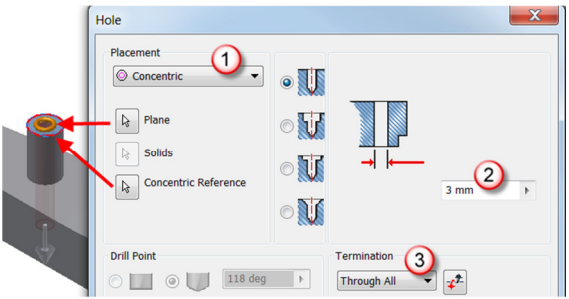
- a) Change distance to **10 mm**.



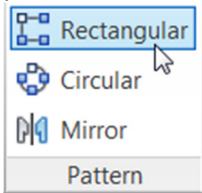
- b) Complete the edit by clicking the green check mark on the mini-toolbar.
- 7) Next place a hole for the axle. Start the Hole command by right-clicking on a blank area in the graphics window and click Hole on the marking menu, or press the H key, or click 3D Model tab > Modify panel > Hole.



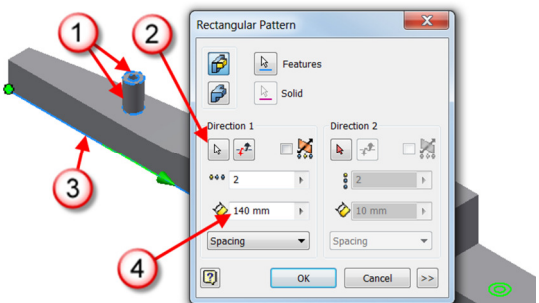
- a) In the Hole dialog box change the Placement option to Concentric labeled (1)
  - i) Define the plane the hole will be placed on by selecting the top planar face of the extruded circle.
  - b) Define what the hole will be concentric to by selecting the outside circular edge or the circular face of the extruded circle.
- c) In the Hole dialog box change the diameter to **3 mm**, labeled (2).
- d) In the Hole dialog box change the Termination to Through All, labeled (3).
- e) In the Hole dialog box, click OK.



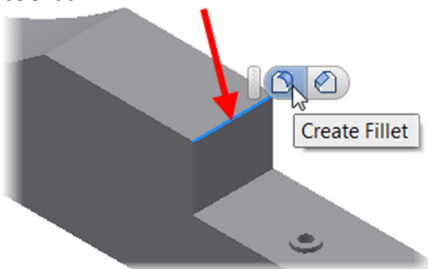
- 8) Next you pattern the wheel extension. Click the Rectangular command on the 3D Model tab > Pattern panel.



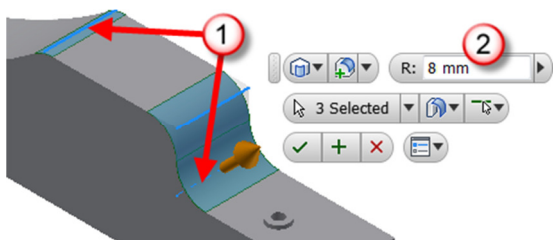
- First select the features to pattern, in the graphics window or in the browser select the circular extrusion and the hole you created in the last steps, labeled (1).
- Next you define the direction of the pattern. In the Rectangular Pattern dialog box select the Direction 1 arrow, labeled (2) and then select the bottom edge on the car body, labeled (3).
- Change the spacing distance to **140 mm**, labeled (4).
- Click OK



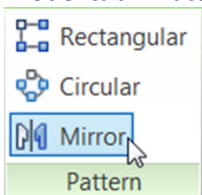
- 9) Next you add a few fillets to the body. Start the Fillet command by selecting the vertical edge on the body and click Create Fillet from the mini-toolbar.



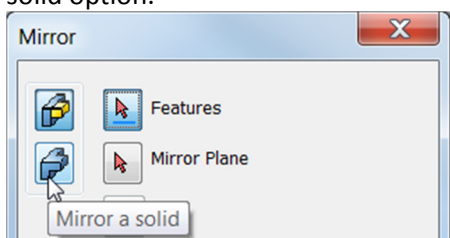
- Select the two additional edges, labeled (1)
- In the mini-toolbar change the radius to **8 mm**, labeled (2).
- To create the fillets, click the green check mark in the mini-toolbar.



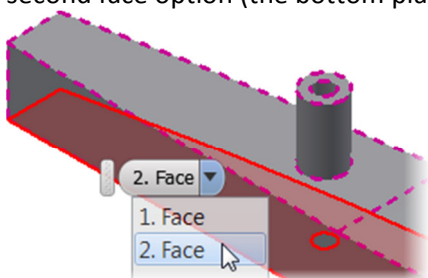
10) Next you mirror the body to create the other half of the car. Start the Mirror command on the 3D Model tab > Pattern panel.



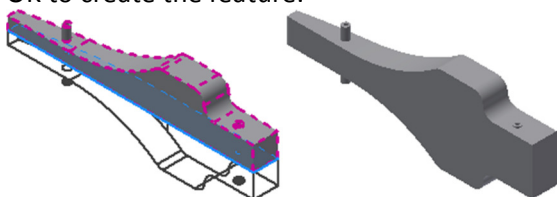
a) In the Mirror dialog box, select the Mirror a solid option.



b) Next you select the plane to mirror the body about, move the cursor over the front vertical plane and let the cursor stay still for two seconds and the Select Other toolbar appears, click the drop down arrow and click the second face option (the bottom plane).



c) After the preview of the mirror appears, click OK to create the feature.



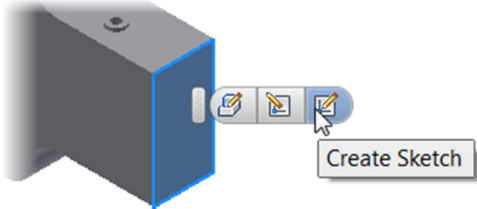
11) Save the file.

### Exercise 1: Section 3 – CO2 Cartridge and Wings

Continue working on the same file or if you were not able to complete the previous section, open the file *Ex1 Sec3 F1 Car Body.ipt*.

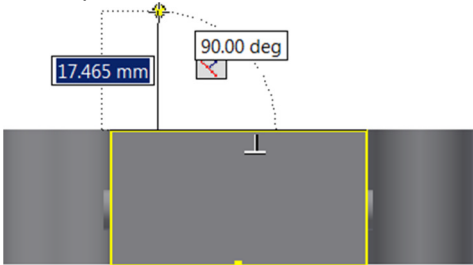
1) Next you create the feature that will contain the CO2 cartridge. Start by creating a new sketch by clicking on the front-side planar face and click

Create Sketch on the mini-toolbar.

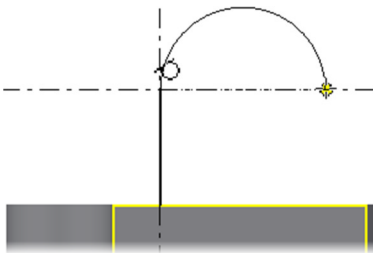


- 2) Sketch the profile that defines the shape that will be extruded, right-click on a blank area in the graphics window and click Create Line on the marking menu.

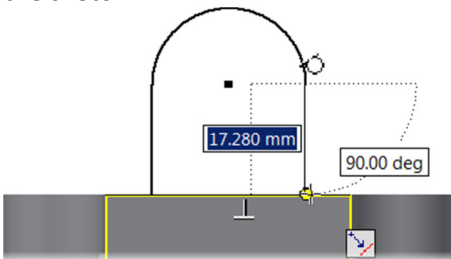
- a) Place the first point of the line on the left side sketch and then click a point above to create a perpendicular line, no dimension will be created at this point (the length of the line is not important as it will be defined later).



- b) While still in the line command, click on the top endpoint of the line and a small gray circle will appear. Click on the small circle, and with the left mouse button pressed down, move the cursor up and to the right until it touches the horizontal centerline line and click.



- c) Complete the profile by clicking a point on the top horizontal edge so the line is tangent to the arc and perpendicular to the top edge of the sketch.

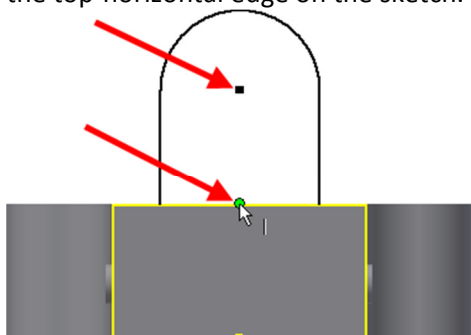


- d) Finish the line command by right-clicking and click OK in the marking menu.
- 3) The sketch should require three dimensions or constraints to be fully constrained. First you center the profile by applying a Vertical sketch constraint, click the Sketch tab > Constrain panel

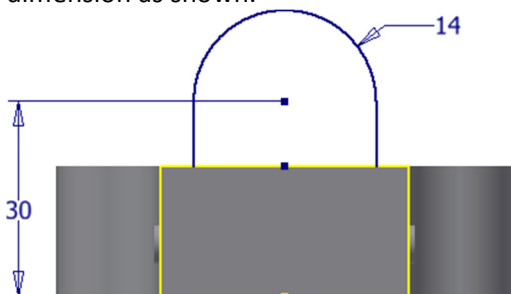
> Vertical Constraint.



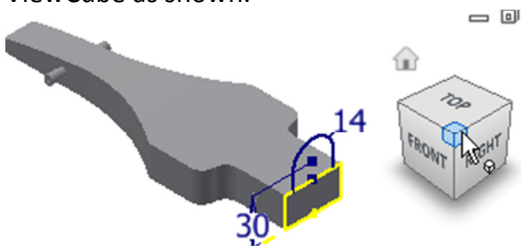
- a) Align two points vertically by clicking the center point of the arc and the midpoint of the top-horizontal edge on the sketch.



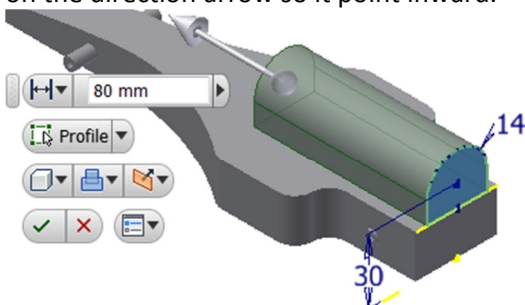
- b) Start the dimension command and add a **30 mm** vertical dimension and a **14 mm** radial dimension as shown.



- c) Finish the sketch.  
4) Change the viewpoint by clicking the corner of the ViewCube as shown.



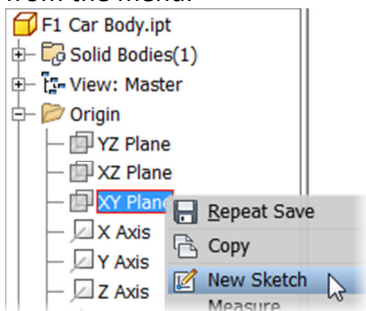
- 5) Extrude the profile **80 mm**. You will need to change the direction of the extrusion by either selecting the Direction 2 option or click and drag on the direction arrow so it point inward.



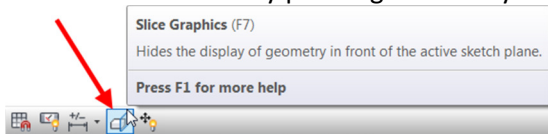
- 6) Next you create the front and rear wings. The profiles of the wings will be sketched in the

middle of the car and then extruded the profiles symmetrically.

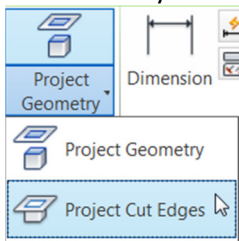
- a) In the browser, expand the Origin folder and right-click on XY Plane and click New Sketch from the menu.



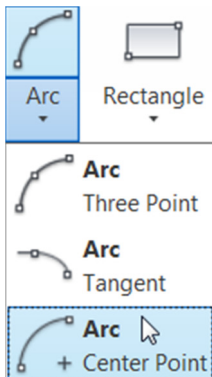
- b) The problem is that the sketch is in the middle of the part and you cannot see the sketch. Remove the front section in front of the sketch by clicking the Slice Graphics command on the Status Bar or by pressing the F7 key.



- c) The front wing will consist of an arc that is centered on the hole of the axle and will use the left vertical edge and bottom horizontal edge to close the profile. To copy this geometry onto the active sketch click the Project Cut Edges command on the Sketch tab > Draw panel > select the down arrow next to Project Geometry > Project Cut Edges. The edges that touch the active sketch will automatically be copied onto the sketch.

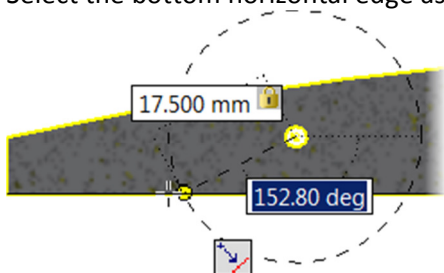


- d) Sketch outline both Front using the Arc Center Point command, click the Sketch tab > Draw panel > select the down arrow below Arc > click Arc Center Point.

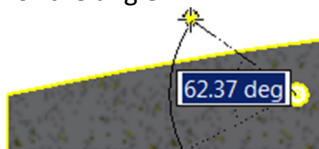


- e) Select the center point of the hole

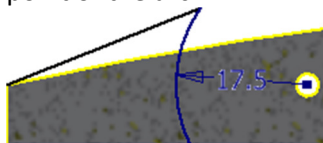
- f) Enter **17.5** for the radius and press the Tab key.
- g) Select the bottom horizontal edge as shown.



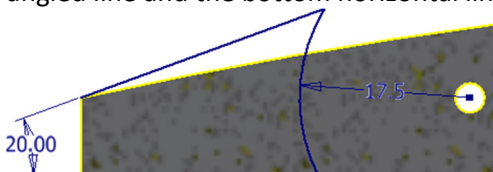
- h) For the last point, click a point above the second point as shown. Do not enter a value for the angle.



- i) Start the line command and select the top endpoint of the left vertical edge and the top point of the arc.

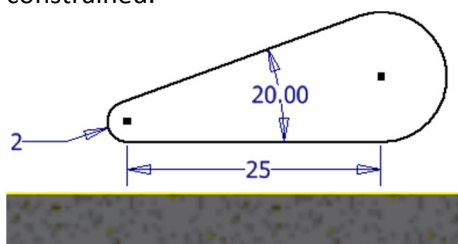


- j) Add a 20 degree dimension between the angled line and the bottom horizontal line.



- 7) Next you sketch the profile for the rear wing. To avoid repeating steps you will use the existing sketch.

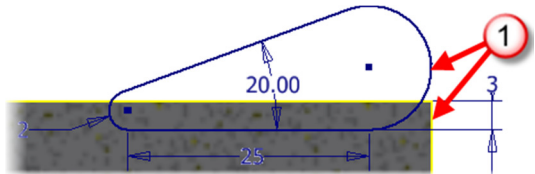
- a) Sketch the profile for the rear wing as shown. For clarity, draw the sketch above and to the right of the sketch. If needed add a Tangent constraint between the arcs and lines. When done the sketch should require two dimensions or constraints to be fully constrained.



- b) Click and drag the sketch so the horizontal line is within the part.
- c) Add a tangent constraint between the right vertical edge and the right arc, labeled (1).
- d) Add a **3 mm** vertical dimension between the top edge and the horizontal line you just



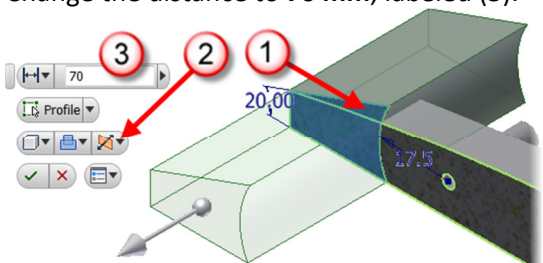
drew.



- 8) Change to an isometric view by click on the upper right corner of the ViewCube as shown.



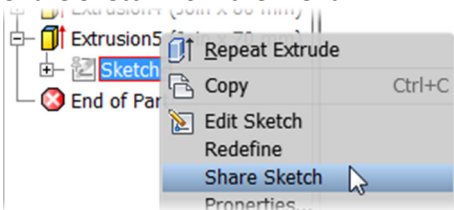
- 9) Next you extrude the front wing, while still in the sketch environment which keeps the graphics sliced.
- Press the E key to start the Extrude command.
  - Define the profile by selecting inside the sketch of the front wing, labeled (1).
  - In the mini-toolbar change the direction to Symmetric, labeled (2).
  - Change the distance to **70 mm**, labeled (3).



- e) Click the green check mark to create the extrusion.

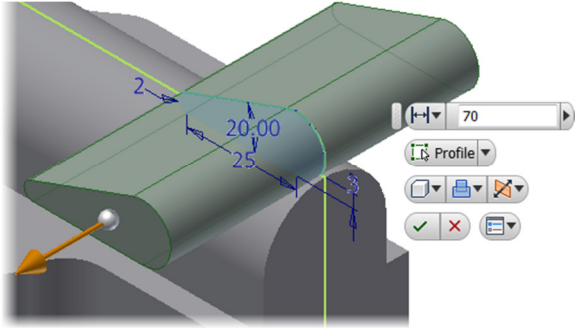
**Note: Both the front and rear wings could have been extruded to create a single feature, but by keeping them separate, you can edit their extrusion distance independently.**

- 10) Next you create the rear wing, using the sketch geometry you created in a previous step. To use an existing sketch, you share it. In the browser expand the last Extrusion by clicking the + and then right-click on the sketch entry and click Share Sketch from the menu.

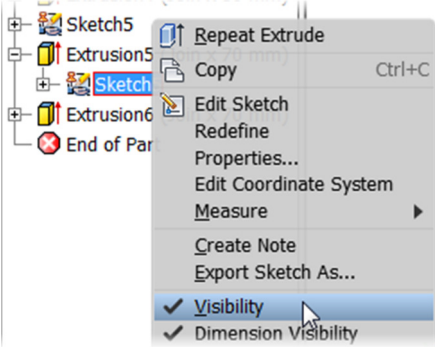


- 11) Start the Extrude command by clicking on any geometry in the sketch and click Create Extrude from the mini-toolbar, or press the E key or click 3D Model tab > Create panel > Extrude and extrude the rear wing sketch **70 mm** with the symmetric direction like you did in the previous

step.

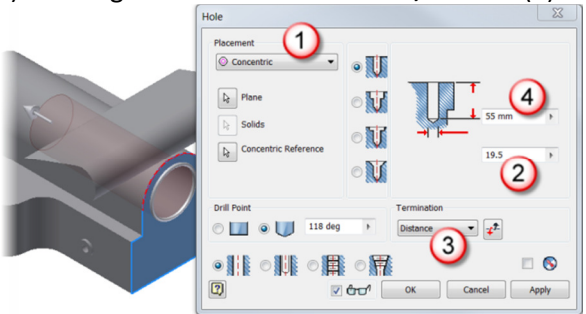


- 12) You know longer need to see the shared sketch, in the browser, right-click on the shared sketch and click Visibility from the menu.



- 13) Next place a hole for the CO2 canister.

- Start the Hole command
- Change Placement to Concentric, labeled (1).
- Select the back plane and circular edge on the car body.
- In the Hole dialog box change the diameter to **19.5 mm**, labeled (2).
- Change the Termination to Distance, labeled (3).
- Change the distance to **55 mm**, labeled (4).



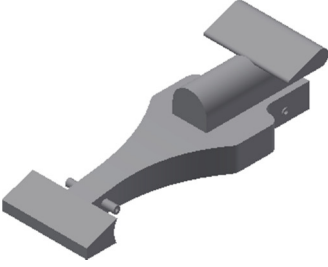
- 14) Save the file.

### Exercise 1: Section 4 – Final Features

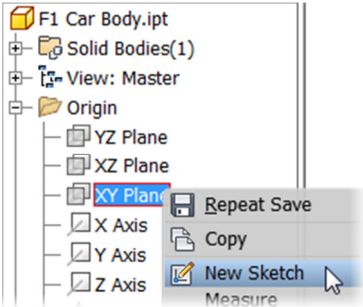
Continue working on the same file or if you were not able to complete the previous section, open the file *Ex1 Sec4 F1 Car Body.ipt*.

- In this step you angle the front face of the CO2 cartridge holder. Change the viewpoint so it

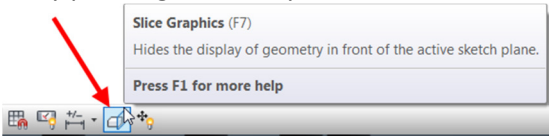
resembles the following image.



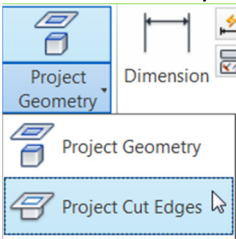
- a) Next you create another sketch in the middle of the car where a profile of the cut will be placed. In the browser, expand the Origin folder and right-click on XY Plane and click New Sketch from the menu.



- b) The sketch again is in the middle of the part and you cannot see the sketch. Remove the front section in front of the sketch by clicking the Slice Graphics command on the Status Bar or by pressing the F7 key.



- c) The profile of the cut will utilize the existing left vertical and top horizontal edge of the feature that will hold the CO2 cartridge. To copy this geometry onto the active sketch click the Project Cut Edges command on the Sketch tab > Draw panel > select the down arrow next to Project Geometry > Project Cut Edges. The edges that touch the active sketch will automatically be copied onto the sketch.



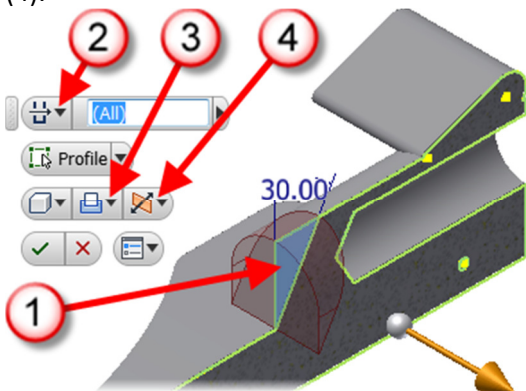
- d) Draw a line that starts at the lower left corner of the left vertical edge on the CO2 cartridge holder, labeled (1).
- e) Click a point on the top horizontal edge of the CO2 cartridge. Labeled (2). Do not place a dimension.

- f) Add a **30 deg** dimension to the left vertical edge and the angled line, labeled (3).

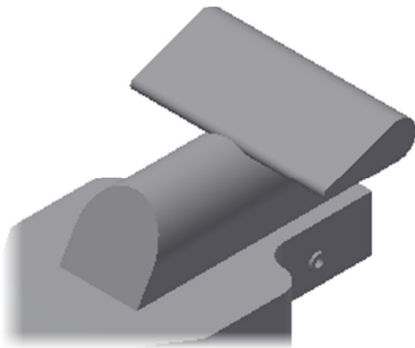


- 2) Next you cut away material to create the angle. While still in the sketch environment press the E key to start the Extrude command.

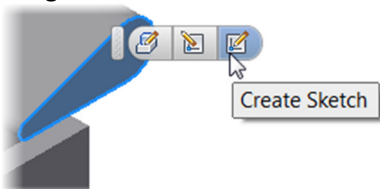
- Define the profile by selecting inside the sketch of the triangle, labeled (1).
- In the mini-toolbar change the Extents option to Through All (2).
- Change the Operation option to Cut (3).
- Change the direction to Symmetric, labeled (4).



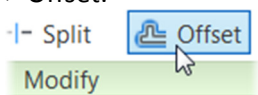
- e) Click the green check mark to create the extrusion.



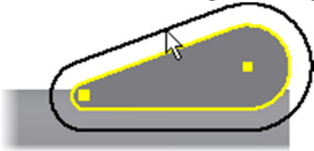
- 3) Next you create a side plate on the rear wing. Create a sketch on the side plane on the rear wing.



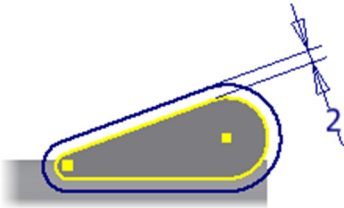
- a) Instead of sketching geometry, you can offset geometry. Click the Sketch tab > Modify panel > Offset.



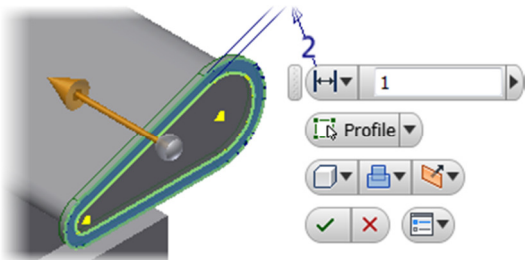
- b) Select the angle line on the sketch and move the cursor out away from the line and click to create the offset geometry.



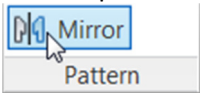
- c) Start the Dimension command and select the two angled lines and place a **2 mm** aligned dimension.



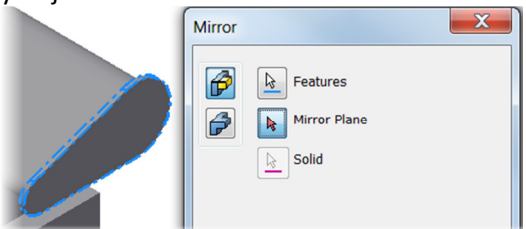
- 4) If needed change the viewpoint so it resembles the following image.
- 5) Use the Extrude command to create the side plate.
- For the profile, select between the offset geometry and the original sketch geometry.
  - Change the direction so it point into the car.
  - Change the extrusion thickness to **1 mm**.
  - Change the operation to Join.
  - Create the feature.



- 6) Next mirror the side plate to the other side by using the Mirror command on the 3D Model tab > Pattern panel.

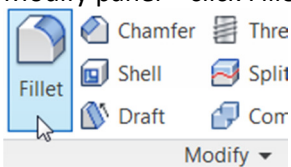


- a) For the feature to mirror select the feature you just created.

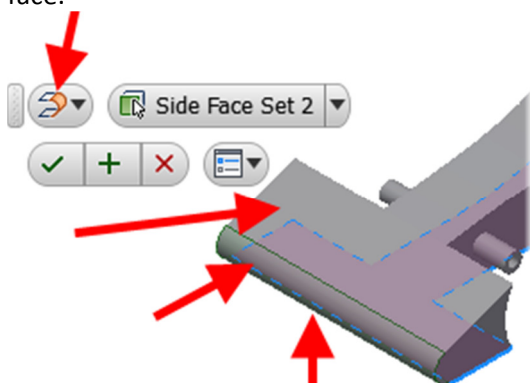


- In the Mirror dialog box click the Mirror Plane button.
- In the browser, under the Origin folder, select XY Plane
- Complete the command by clicking the OK button in the Mirror dialog box.

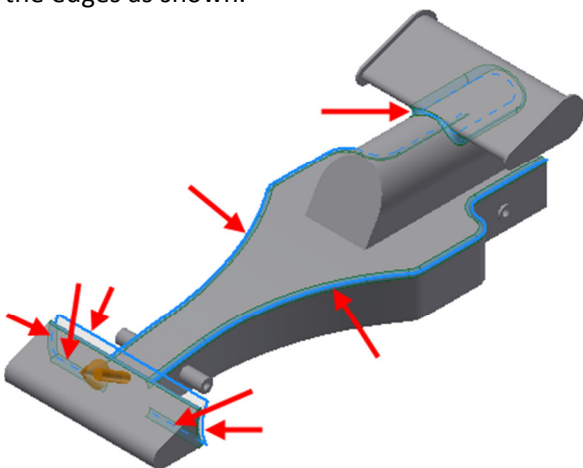
- 7) Next you round off the front wing. It is best practice to apply fillets near the end of the modeling process as the edges will be consumed. Press the F key or from the 3D Model tab > Modify panel > click Fillet.



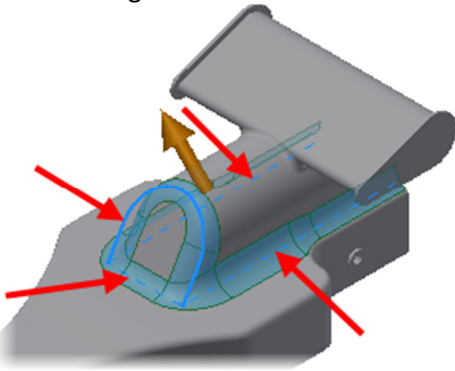
- On the mini-toolbar click the Full Round Fillet option.
- Select the faces that the fillet will be tangent to, in order select the top angled face, the front vertical face and the bottom horizontal face.



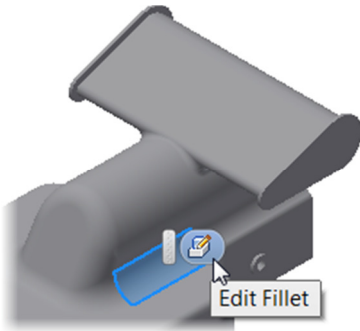
- In the Fillet dialog box or on the mini-toolbar, click OK to create the fillet.
- 8) Use the Fillet command to place **2 mm** fillets to the edges as shown.



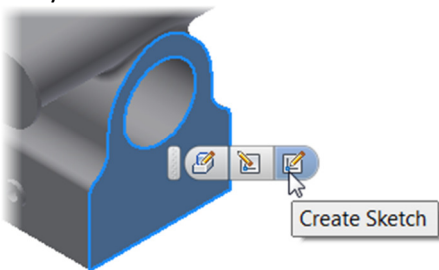
- 9) Use the Fillet command to place **7 mm** fillets to the four edges as shown.



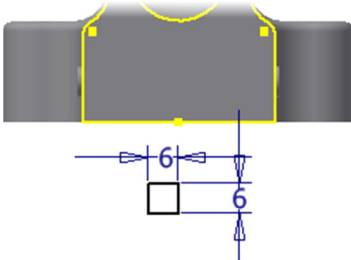
- 10) Practice changing the size of the fillets by selecting a fillet in the graphics window and click Edit Feature from the mini-toolbar.



- 11) Next you create a cutout for the tether line.  
a) Create a sketch on the back face of the car body.

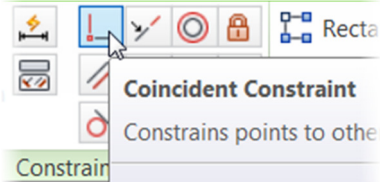


- b) Use the Rectangle command to draw a **6mm** square below the car body. It is placed below the car body for clarity.

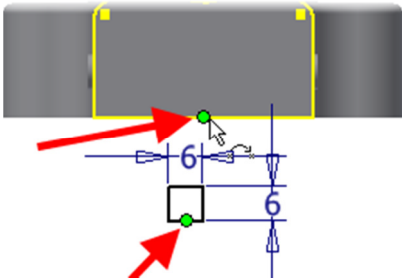


- c) Next you constrain the sketch by applying a coincident constraint that will lock two points together. Apply a Coincident sketch constraint by clicking the Sketch tab >

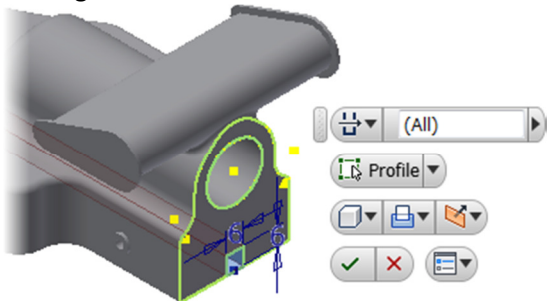
Constrain panel > Coincident Constraint.



- d) Select the two midpoints on the bottom horizontal line of the square and the midpoint of the bottom horizontal edge of the car body.

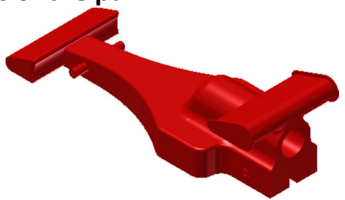
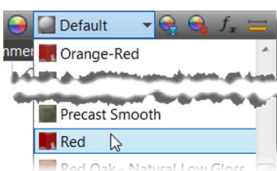


- e) Extrude the square profile with the cut operation and the Extents option set to Through All.



- 12) Next you change the appearance / color of the car by selecting an appearance override from the Quick Access toolbar, to the right of the list of Materials. Select an appearance / color of your choice.

**Note: The appearance override does not affect the mass properties of the part**



- 13) Save the file.

- 14) If desired open the completed car, *F1 Car Body Completed.ipt*.

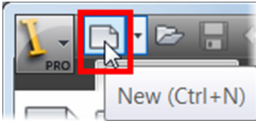


## Exercise 2: Section 1 – Creating Drawing Views

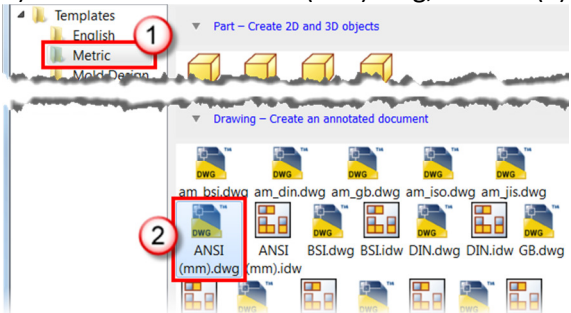
In this exercise, you create a drawing of the F1 car body so the car can be made.

**Note: Depending upon the drafting standard you use, you may choose a different template file. The ANSI mm template file used third angle projection.**

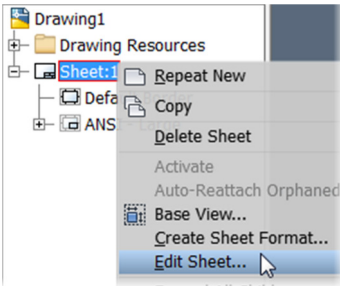
- 1) Start by creating a new drawing file. Click the New command on the Quick Access toolbar.



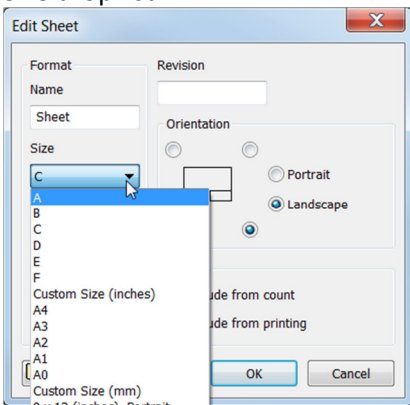
- a) In the Create New File dialog box click the Metric folder under Templates, labeled (1).
- b) Double-click on ANSI (mm).dwg, labeled (2).



- 2) If needed you can change the sheet size by click Edit Sheet in the browser.

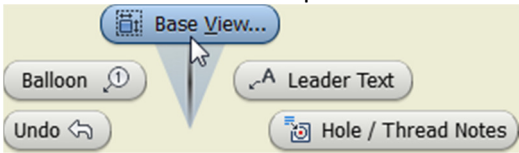


- a) In the Edit Sheet dialog box select the desired sheet size by selecting a sheet size from the Size drop list.

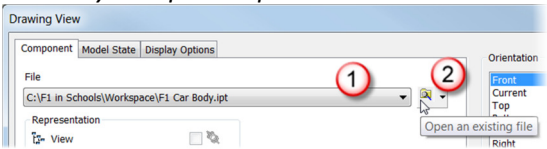


- b) For this exercise, you use the default C size, click the Cancel button to close the dialog box without making changes,
- 3) The next step is to place drawing views; orthographic and isometric views by using the Base view command. Click the Base View command by either , right-clicking on a blank area in the graphics window and click Base

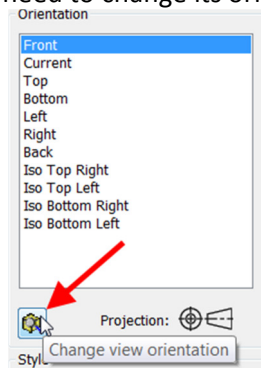
View on the marking menu, or click Base from the Place Views tab > Create panel.



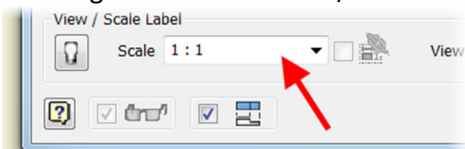
- a) In the Drawing View dialog box, select the file to create the views from, if the file is open you can select it from the list labeled (1) in the following image or select the Open an existing file button labeled (2) in the following image. If you completed the previous exercise select the file *F1 Car Body.ipt* or if you did not complete the last exercise select the file *F1 Car Body Completed.ipt*.



- b) In the graphics window a preview of the drawing view will be displayed, you can adjust the view's orientation by selecting a view from the Orientation list or adjust the view manually by clicking the Change view orientation option. For this exercise the default Front view is correct so you do not need to change its orientation.

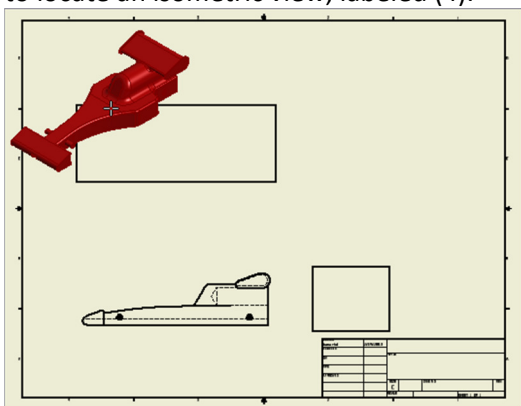


- c) If needed you could also adjust the scale of the drawing view by selecting a scale or entering a scale in the View / Scale Label area.

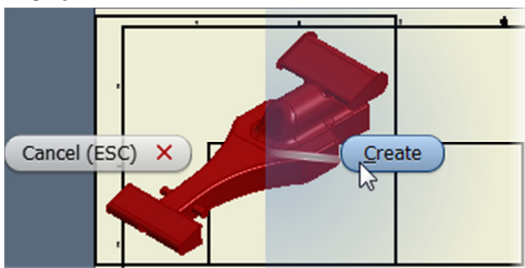



- d) Move the Drawing View dialog box to the side and click to locate the base view (first view) near the lower left corner of the drawing view, labeled (1).
- e) Move the cursor up and click to locate the top view, labeled (2).

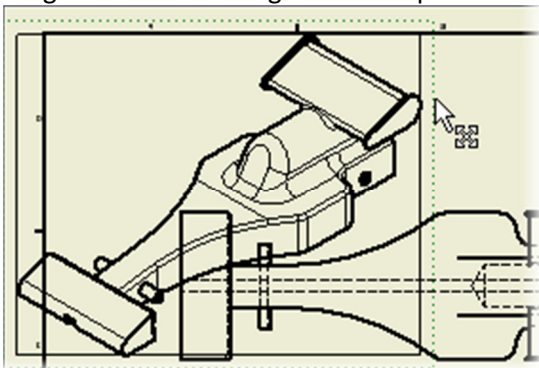
- f) Move the cursor to the right and click to locate the right side view, labeled (3).
- g) Move the cursor up and to the left and click to locate an isometric view, labeled (4).



- h) The last step is to create the views by right-clicking and click Create from the marking menu.

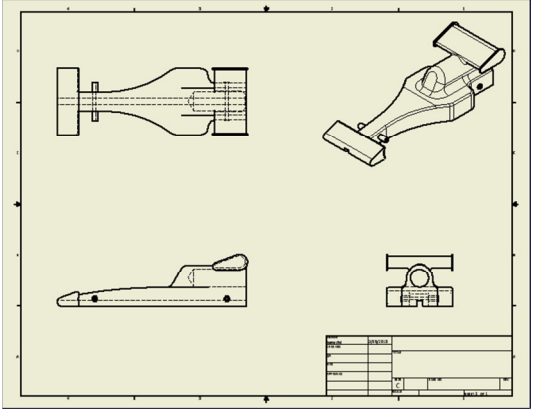


- 4) Next you move the drawing views to better fit the drawing. Move the cursor over the isometric view until the  symbol appears and then click and drag the view to the right of the top view.



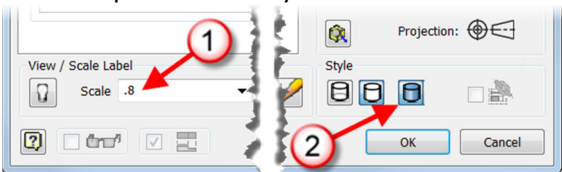
- a) Next move the base view (first view placed, which in this exercise is the front view) down and to the left. Notice that the orthographic views (top and side views) will move with it.

- b) Position all the view as shown.



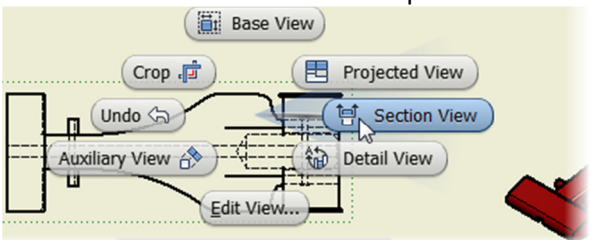
- 5) Next you adjust the scale and style of the isometric view. Edit the isometric view by double-clicking in the isometric view

- a) Near the bottom of the Drawing View dialog box change the scale by entering **.8** in the View / Scale Label area, labeled (1).
- b) While still in the Drawing View dialog box change the style of the view by clicking the Shaded option in the Style area.

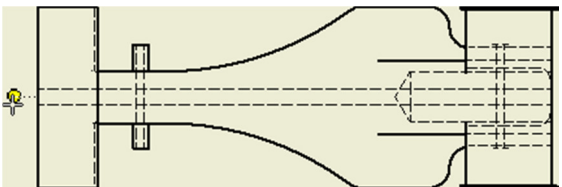


- c) Click OK to complete the edits.

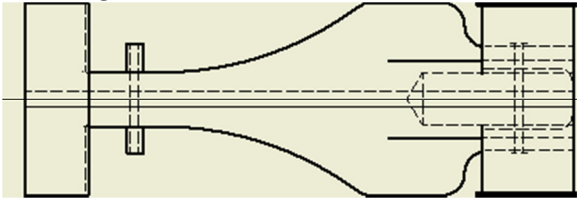
- 6) To see the inside of the middle of the car, you can create a section view. In this step you create a section view based on the top view. Start the Section View command by either right-clicking in the top view and click Section View from the marking menu or click the Section View command from the Pace Views tab > Create panel.



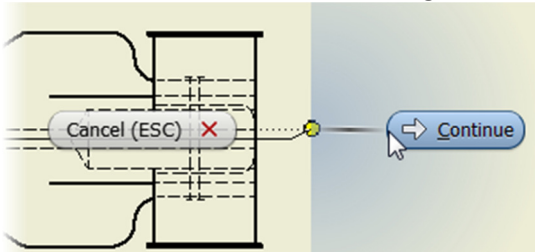
- a) If you started the command from the Create panel you need to select the view to section, click inside the top view, if you started the command by right-clicking in the top view the view is already selected.
- b) First you locate the first point of the section line by clicking a point to the left and in the middle of the left vertical line.



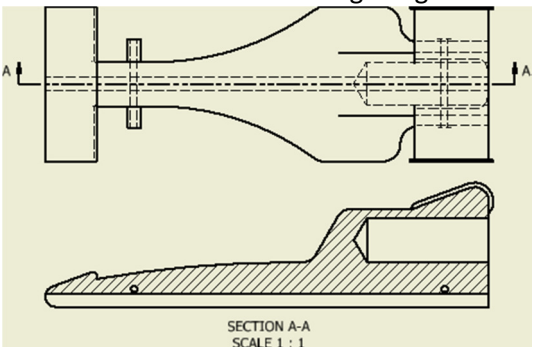
- c) Next you define the section line view by clicking a point to the right and in the middle of the right vertical line.



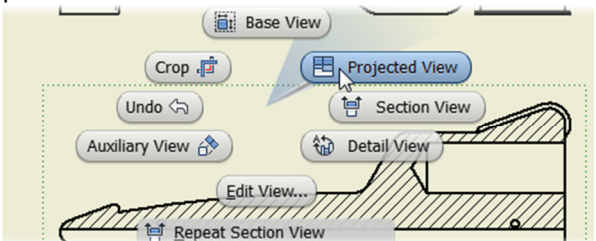
- d) You can continue to define points of the section but in this case a single section line is needed. To move to the next task, right-click and click Continue from the marking menu.



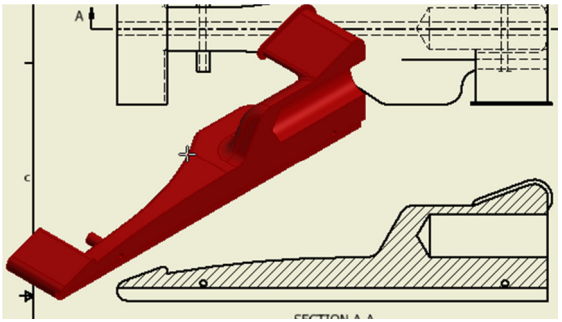
- e) Position the section view by clicking a point below the top view. When done your view should resemble the following image.



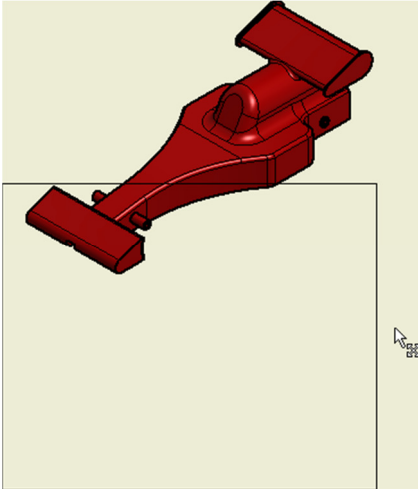
- 7) To see the cross section view from an angle you create an isometric view by using the Projected View command. To create an isometric view of the cross section view, right-click in the cross section view and click Projected View from the marking menu. You can also access the Projected View command from the Place Views tab > Create panel.



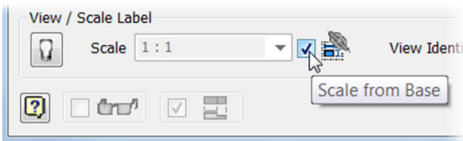
- a) Locate the view by clicking a point above and to the left of the cross section view.



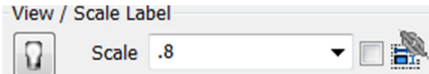
- b) Move the isometric of the cross section view so it is below the existing isometric view.



- c) Next you change the scale of the new isometric view by double-clicking in the isometric view. Since this view was projected the scale area is grayed out. To change the scale, uncheck the Scale from Base.



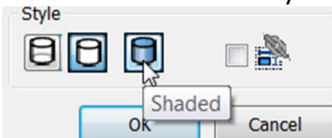
- d) Then enter .8 for the scale.



- e) While still in the Drawing View dialog box, change the style to Shaded, this view's style is currently linked to the cross section view. To change the style, first uncheck the Style from Base.

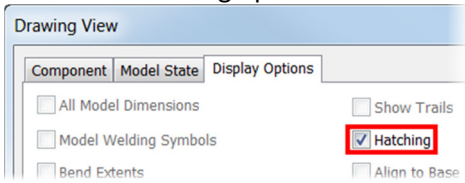


- f) Select Shaded as the style.

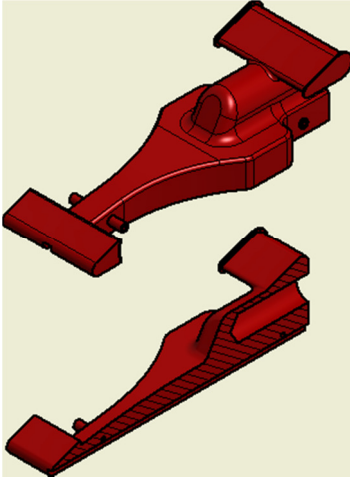


- g) While still in the Drawing View dialog box, you turn on the hatching option for this view. Click

the Display Option tab in the dialog box and check the Hatching option.



h) Click OK and then position the view as shown.

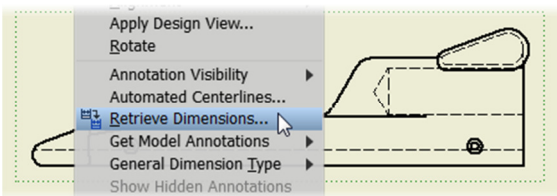


- 8) Save the file with the name *F1 Car Body.dwg*.  
Note: The drawing's name will default to the name of the file that the base view was created from.

### Exercise 2: Section 2 - Drawing Annotation and Edits

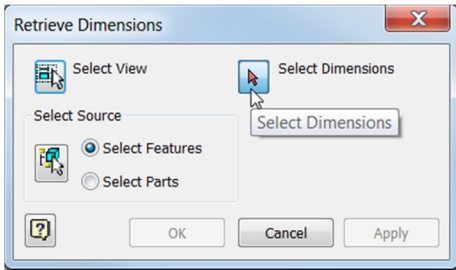
Continue working on the same file or if you were not able to complete the previous section, open the file *Ex2 Sec2 F1 Car Body.dwg*. In this portion of the exercise you add annotations to the drawing views and change the size of the car.

- 1) The dimensions that were placed when you created sketches can be retrieved. Use the Retrieve Dimensions command by either right-clicking in a view and click Retrieve Dimensions from the menu or click Retrieve Dimensions from the Annotate tab > Dimension panel.
- a) Retrieve the dimensions in the front view by right-clicking in the front view and click Retrieve Dimensions from the menu.

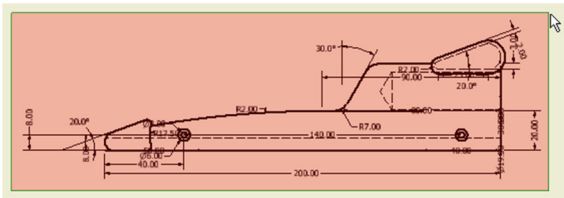


- b) In the Retrieve Dimensions dialog box click the Select Dimensions button, this will display all of the dimensions that are possible to be

retrieved in this view.



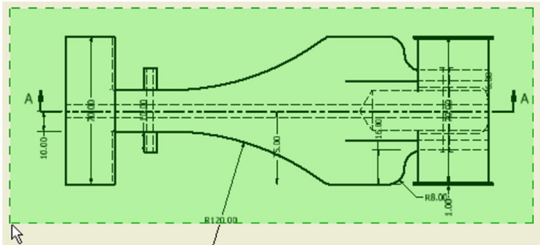
- c) In this step you use the window selection method to select all of the dimensions in the view. The window selection method is when you drag a window from left to right and the entities that are completely enclosed in the window are selected. Select all the dimensions in the front view by clicking a point that is below and to the left of the of all the dimensions that are in the front view and with the left mouse button pressed, drag a window up and to the right so all the dimensions are contained in the window and then release the mouse button.



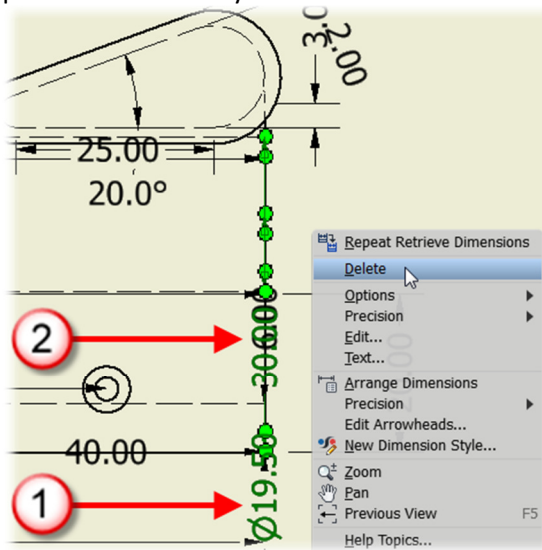
- d) Retrieve the dimensions by clicking Apply in the dialog box, this keeps the command active.
- 2) With the Retrieve Dimensions dialog box still open, click in the top view.
- a) In the Retrieve Dimensions dialog box click the Select Dimensions button, this will display all of the dimensions that are possible to be retrieved in this view.
- b) In this step you use the crossing selection method to select all of the dimensions in the view. The crossing selection method is when you drag a window from right to left and the entities that are completely enclosed and the entities that touch the selection box are selected. Select all the dimensions in the top view by clicking a point that is above and to the right of the of all the dimensions that are in the top view and with the left mouse button pressed, drag a window down and to the left so all the dimensions are contained in the window and then release the mouse



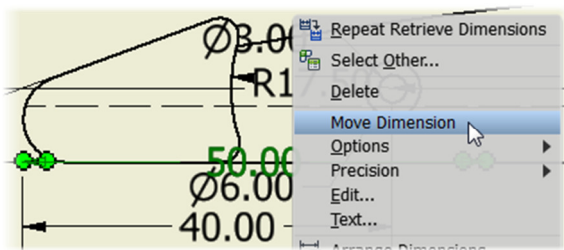
button.



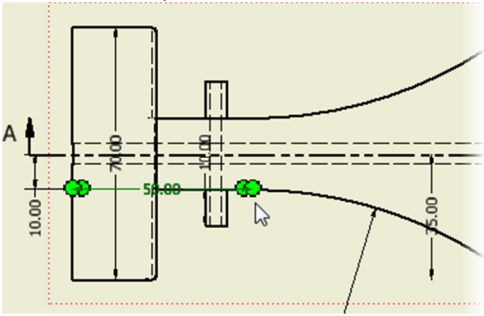
- c) Retrieve the dimensions by clicking Apply in the dialog box.
- 3) With the Retrieve Dimensions dialog box still open retrieve all the dimensions in the side view using either the window or crossing selection method.
  - a) Click OK to complete the command.
- 4) You can delete dimensions that are not needed by selecting them and then either press the Delete key or right-click and click Delete from the menu. In the front view, select the 19.50 diameter dimension, labeled (1), hold down the Ctrl key and then select the 30.00 dimension, labeled (2) and right-click and click Delete from the menu or press the Delete key.



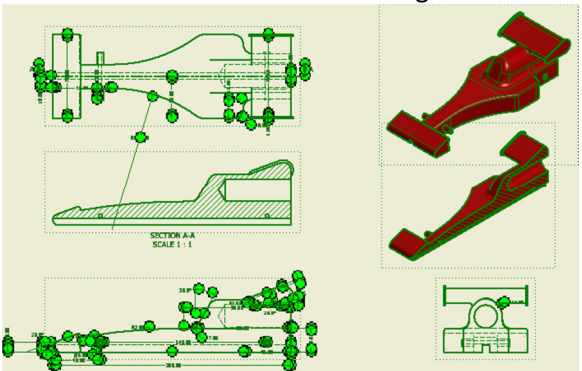
- 5) You can also move a dimension between views by using the Move Dimension command. Move the cursor over the 50.00 dimension in the front view (do not click on the dimension) and click Move Dimension from the menu.



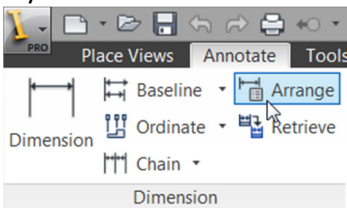
- a) Select the view to move the dimension to, click in the top view.




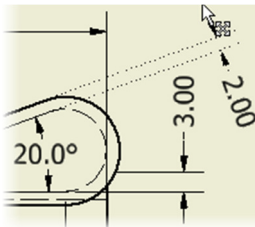
- 6) Next you arrange all of the dimensions in one operation. Use the window or crossing selection technique that was covered previously to select all of the dimensions in the drawing.



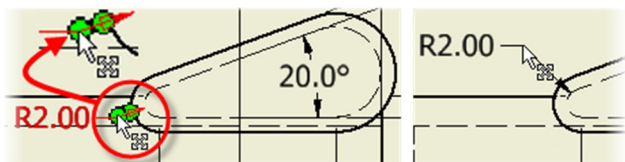
- a) Start the Arrange Dimensions command by clicking the Annotate tab > Dimension panel. The dimensions will automatically be arranged based on the current dimension style.



- 7) Adjust the location of a dimension by moving the cursor over the dimension you want to move until the  symbol appears and then click and drag it to a new location. Move the 2.00 aligned dimension in the front view as shown. Adjust

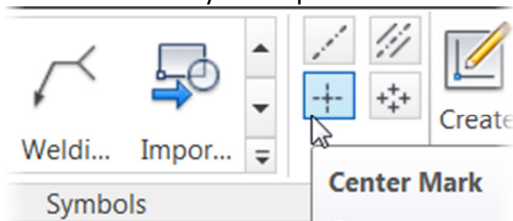


- a) Also in the front view, rotate the 2.00 radial dimension, so it is not inside the geometry. Click and drag on the endpoint of the line with the arrow and drag the dimension up.

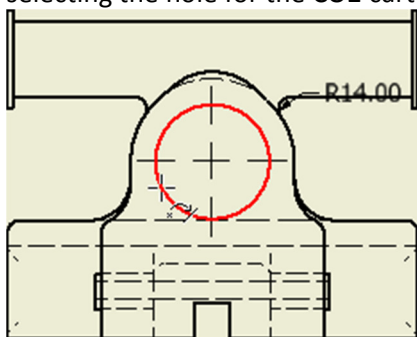


b) Practice moving and deleting dimensions as needed.

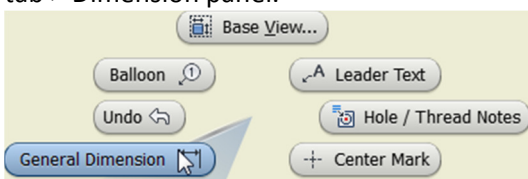
- 8) Next add a center mark to the hole in the right side view. Click the Center Mark command on the Annotate tab > Symbols panel.



a) Create a center mark in the right side view by selecting the hole for the CO2 cartridge.

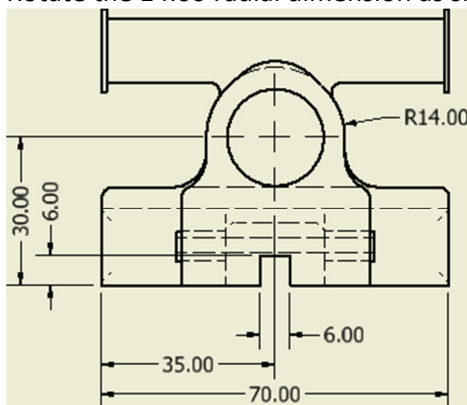


- 9) There may be times that you want to place additional dimension to clarify the design. To add a dimension either move the cursor so it is outside a drawing view then right-click and click General Dimension from the marking menu or click the Dimension command on the Annotate tab > Dimension panel.



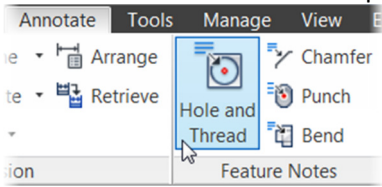
a) In the right-side view add the five linear dimensions as shown.

b) Rotate the 14.00 radial dimension as shown.

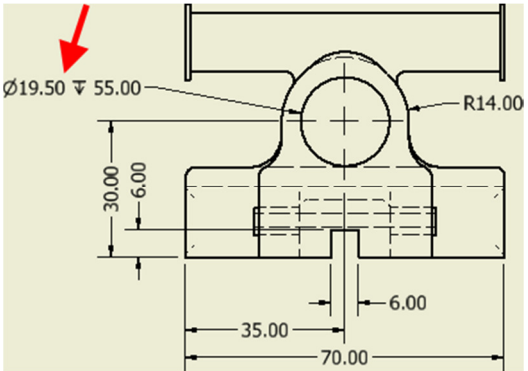


- 10) Next you add a hole note to document the size of the hole that will contain the CO2 cartridge. Start

the Hole and Thread Notes command on the Annotate tab > Feature Notes panel.

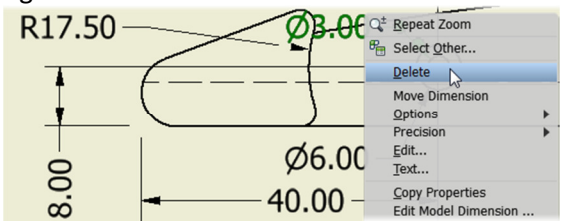


- a) Select the hole in the side view and then click to locate the hole note as shown.

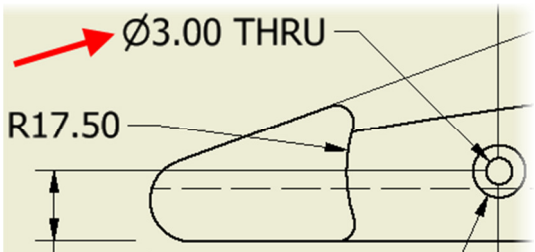


- b) Create the hole note by right-clicking and clicking OK from the marking menu.

- 11) In the front view delete the 3.00 diameter dimension on the hole for the left axle and arrange the dimensions as shown.



- a) Add a hole note to the hole in the front axle.

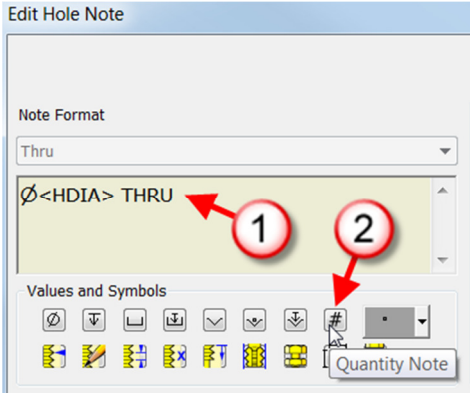


- b) Finish the Hole and Thread Note command.

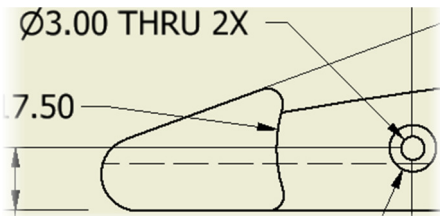
- 12) Notice that the 3.00 THRU hole note does not have a quantity listed, since there are multiple holes with the same size you can add a quantity to the hole note. To add a quantity to the hole note, double-click on the hole note and the Edit Hole Note dialog box appears.

- a) Locate where the quantity note will be placed by clicking after the word THRU and press the Spacebar, labeled (1).

- b) Click the Quantity Note button, labeled (2).



- c) Complete the edit by clicking OK.

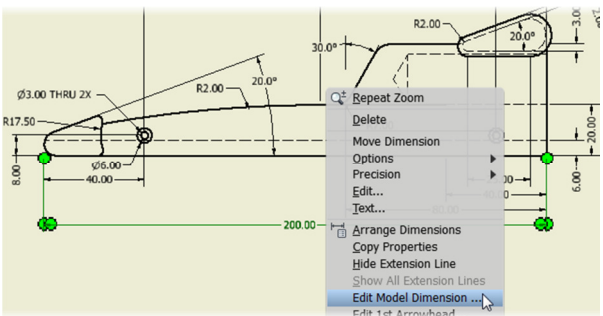


- d) Save and then close the file.

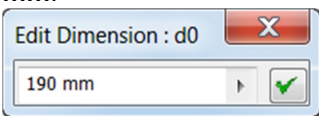
### Exercise 2: Section 3 – Editing the Model

In this section of the exercise you edit the model from within the drawing and from the model.

- 1) Open the file *Ex2 Sec3 F1 Car Body.dwg*. In this file the file *F1 Car Body Completed.ipt* was used.
- 2) While in the drawing you can change the value of a model dimension, as long as the dimension was retrieved, and not placed in the drawing with the General Dimension command. Move the cursor over the 200.00 dimension in the front view, when the dimension is highlighted, right-click and click Edit Model Dimension from the menu.

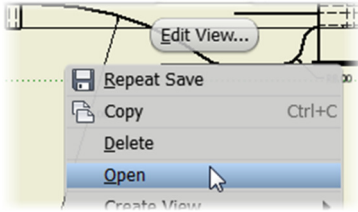


- a) In the Edit Dimension dialog box enter **190 mm**.

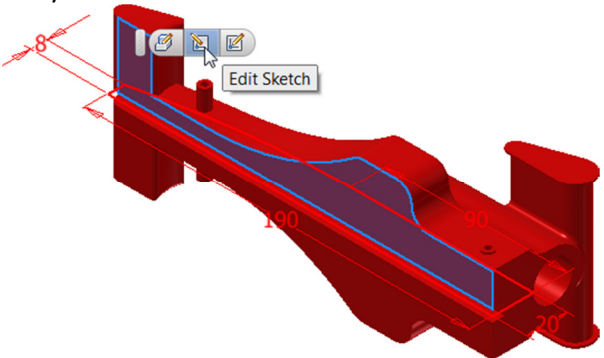


- b) Complete the edit by clicking the green check mark. The part file will update and then the drawing views will update.
  - c) Review the drawing to ensure all of the views were updated to reflect the change.
- 3) Next you edit the same sketch dimension in the part file. Start by opening the part file that the

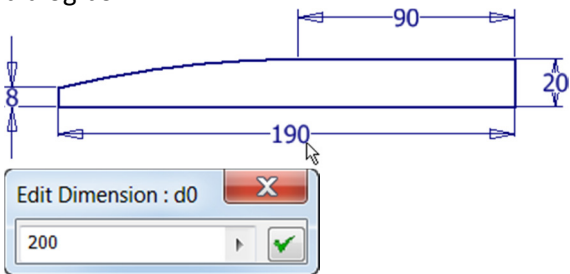
drawing was created from. Instead starting the Open command and navigating to the file, simply right-click in any drawing view and click Open from the menu. Since all the drawing views are based on the same part file, it does not matter which view is used.



- 4) In the part file select the bottom face of the car body and click Edit Sketch on the mini-toolbar.



- a) Edit the 190 mm dimension by double-clicking on it and enter **200 mm** in the Edit Dimension dialog box.

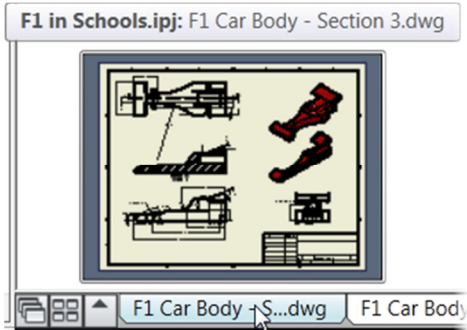


- b) Complete the edit by clicking the green check mark. The part file will update and then the drawing views will update.  
 c) Exit the sketch environment by right-clicking on a blank area in the graphics window and click Finish 2D Sketch from the marking menu.



- 5) Save the file.  
 6) Make the *F1 Car Body Section 3.dwg* drawing current by clicking its tab on the bottom left

above the Status Bar.

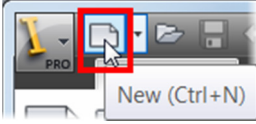


- a) Notice that the dimension in the front view and other views were updated to reflect the change.
- 7) Save and close all Inventor files

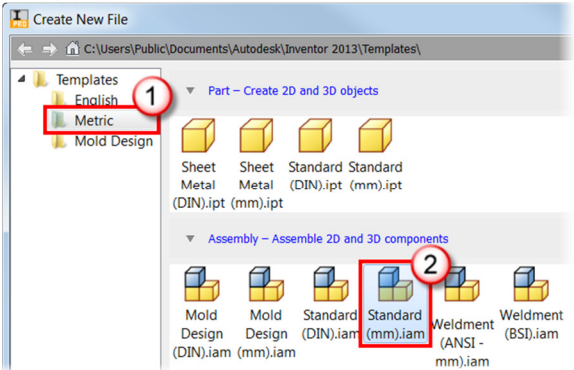
### Exercise 3 – Assembling an F1 Car

In this exercise you assemble the components of an F1 | Schools race car.

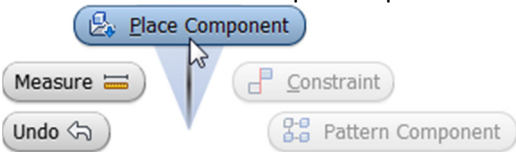
- 1) Start by creating a new assembly file. Click the New command on the Quick Access toolbar.



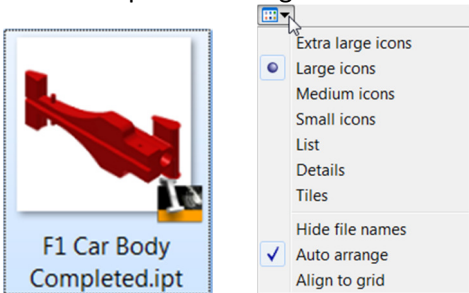
- a) In the Create New File dialog box click the Metric folder under Templates, labeled (1).
- b) Double-click on Standard (mm).iam, labeled (2).



- 2) In the assembly file, the first thing you need to do is place the components. Place components into the assembly right-clicking on a blank area in the graphics window and click Place Component on the marking menu, or click Place Component on the Assemble tab > Component panel.



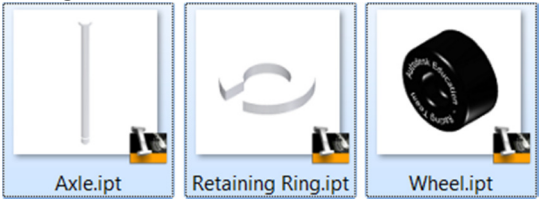
- a) Select the component to place by double-clicking on *F1 Car Body Completed.ipt*. Note that you can change if and how the preview image appears in the icons by selecting an option under the View Menu options in the Place Component dialog box.



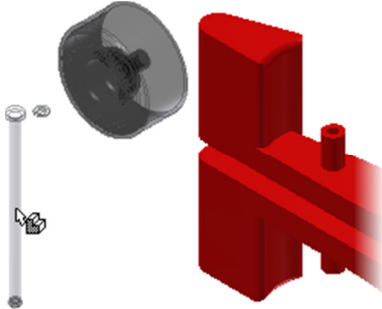
- b) A single instance of the car body will be placed in the assembly. Press the Esc key to cancel the command.
- 3) Next you place the axle and wheel in one operation. Repeat the Place Component command by pressing the Enter key on the keyboard.



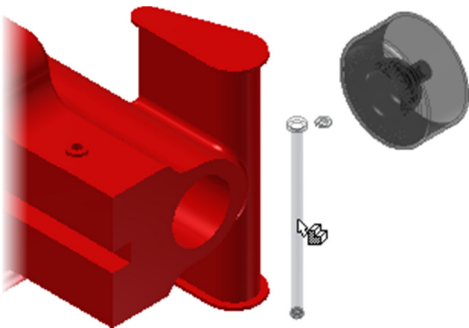
- a) In the Place Component dialog box click once on the *Axle.ipt* and then hold down the Ctrl key and click the *Retaining Ring.ipt*, *Wheel.ipt* files and then click the Open button in the dialog box.



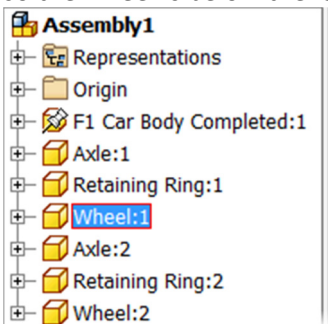
In the graphics window place an occurrence of each of the parts by clicking a point to the left of the front of the car.



- b) Then place a second occurrence of each of the parts by clicking a point to the right of the back of the car.

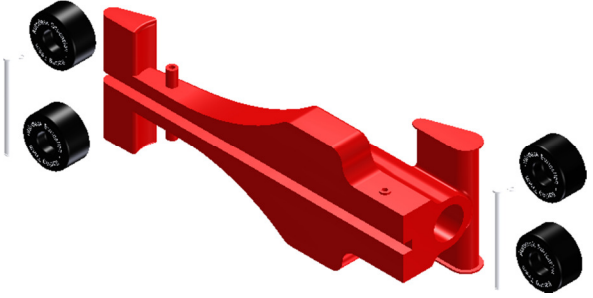


- c) Complete the command by right-clicking and click OK from the marking menu.
- 4) Next you add two more occurrences of the wheel by dragging them from the browser. Move the cursor over *Wheel:1* or *Wheel:2* (the number designates the number of occurrences) and click and drag an occurrence into the graphics window so the wheel is below the left most wheel.

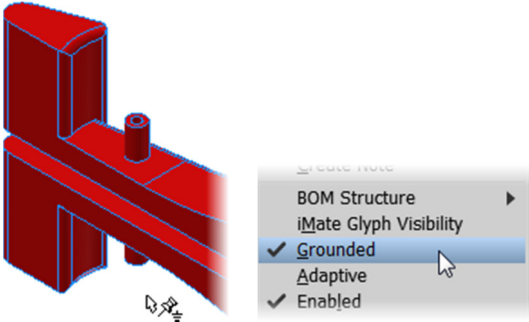


- 5) Repeat the process to drop a fourth occurrence of the wheel into the assembly as shown. If needed

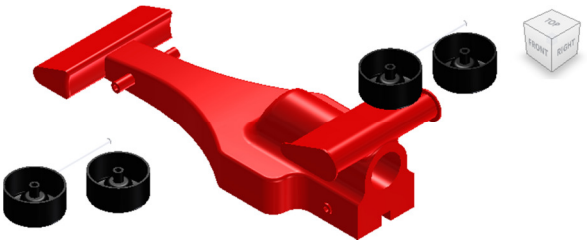
you can click and drag on a wheel to move it.



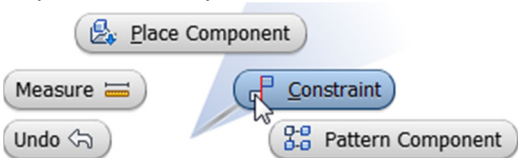
- 6) Save the file as *F1 Race Car Assembly.iam*.
- 7) Next you assemble the race car using assembly constraints. If you are using Inventor 2013 the first part that you place will be grounded, meaning that it does not move. If you are using Inventor 2014, the first placed component is not automatically grounded. Click and drag on the car body, if it is ground a stick pin symbol will appear. If it is not grounded the part will move. If needed, ground the car body by moving the cursor over the car body and click Grounded from the menu.



- 8) Change the viewpoint by clicking the corner of the ViewCube as shown.



- 9) Move a wheel closer to the front left wheel extension and zoom into this area.
- 10) Next you position a wheel on the front wheel extension by applying assembly constraints. Start the Constrain command by right-clicking on a blank area in the graphics window and click Constrain from the menu or click the Constrain command from the Assemble tab > Position panel or press the C key.

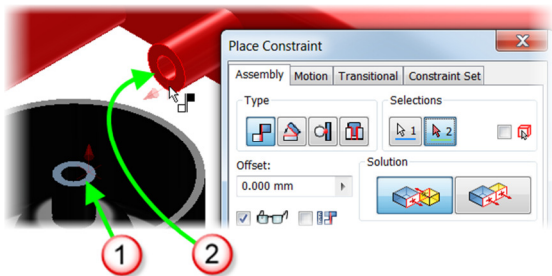


- a) First you mate two planar faces together. Select the planar face of the wheel spacer, labeled (1). You may need to use the select other tool to select the face, with the cursor

positioned over the face for 2 seconds a list will appear and click the Face Normal option.

- b) Then select the outside planar face of the wheel extension, labeled (2).

Note: If needed you can enter an offset value that will move the components away from each other.



- c) Click OK to create the constraint.

**Tip: You do not need to close out of the command between applying constraints, this was done to demonstrate how the Mate constraint works.**

- d) Click and drag the wheel and the wheel moves along the plane that the constraint was created on.
- e) Change the viewpoint as shown in the following image.
- f) Next you center the wheel on the wheel extension again using the Mate constraint. The Mate constraint is dependent upon the geometry that you select; planar faces, circular faces, points or centerlines. Start the Constrain command again and select the circular face on the hole on the wheel and wheel extension, a centerline will appear.

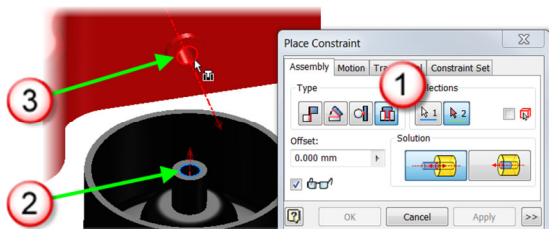


- g) Click OK to create the constraint.
- h) Click and drag the wheel, the wheel rotates along the center of the holes. This is the only open degree of freedom.

- 11) Instead of creating two constraints to position the wheel, you can use the Insert constraint that mates two faces and two centerlines in one operation. Start the Constrain command and click the Insert option, labeled (1).

- a) Select the inside circular edge on the hole of the wheel and the circular edge on the hole in

the wheel extension.



b) Click Apply to create the Insert constraint but keep the dialog box open.

12) Continue to apply insert constraints to the other three wheels, axles and retaining rings. Change the viewpoint and zoom in as needed. You can also exit the Constraint command and move the parts closer together and restart the Constraint command.

**TIP: While in a command you can change the viewpoint by pressing down the F4 key, selecting a corner on the ViewCube, or using the Free Orbit command, and after changing the viewpoint the current command is still active.**

13) Save the file as *F1 Race Car Assembly.iam*

14) Close all Inventor files.

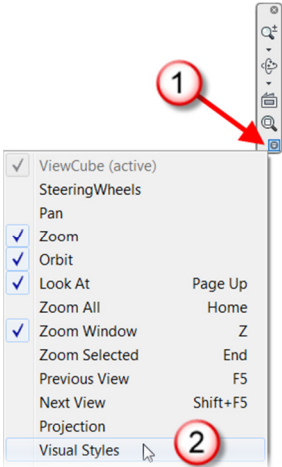
15) If desired examine the completed assembly by opening the file *Ex3 F1 Race Car Assembly Completed.iam*.



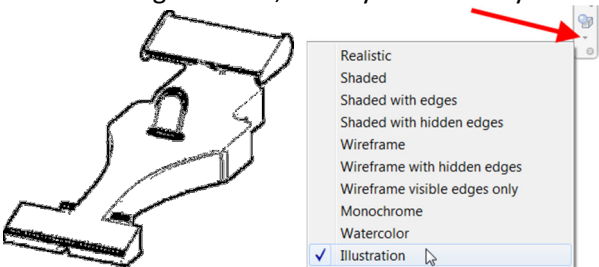
## Exercise 4 – Additional Visualization Options

In this section you learn how to enhance the visualization of the F1 in Schools car by changing visual styles, adding a decal, shadows and reflections, and changing the background and adjust the light source(s).

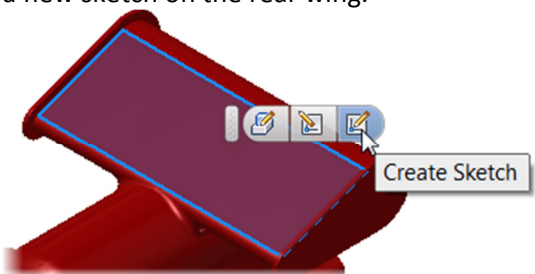
- 1) Open the file *Ex4 F1 Car Body Appearance.ipt*.
- 2) In Exercise 1: Section 4 – Final Features, you learned how to change the appearance / color of the car. In this step you change how the car is displayed on the screen by changing its visual style. The Visual Styles command can be accessed from the View tab > Appearance panel or add it to the Navigation Bar. Add Visual Styles to the Navigation Bar by clicking on the down arrow on the bottom-right corner of the Navigation Bar, labeled (1).
  - a) From the list click Visual Styles, labeled (2).



- b) Click the Visual Styles command on the Navigation Bar, and try different styles.

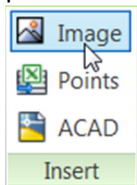


- c) When done change the visual style back to Shaded.
- 2) Next you add a decal to the car. Start by creating a new sketch on the rear wing.



- a) Next insert an image by clicking the Insert Image command on the Sketch tab > Insert

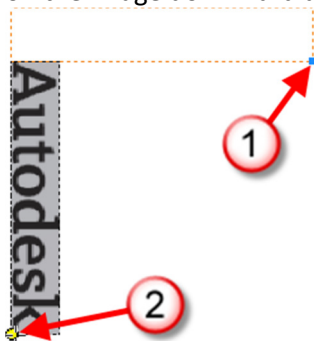
panel.



- b) In the Open dialog box double-click the *Autodesk Logo.png* file.



- c) Place the image by clicking a point to the right of the back of the car.
- d) To rotate the image 90 degrees clockwise, by clicking and dragging the bottom-right point on the image downward as shown.

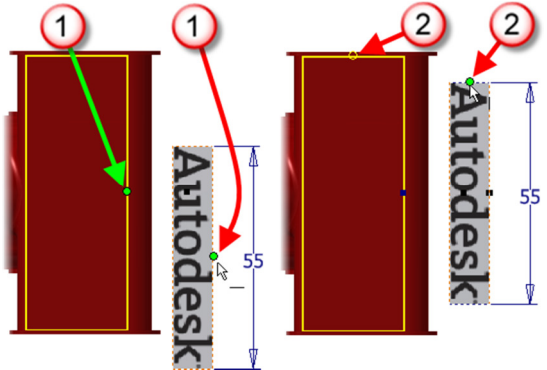


- e) Even though the image is rotated, the image can still rotate. To prevent the image from rotating apply a vertical constraint to one of the longer edges on the image. Start the Vertical constraint from the Sketch tab > Constrain panel.
- f) Add a **55 mm** vertical dimension to the image.



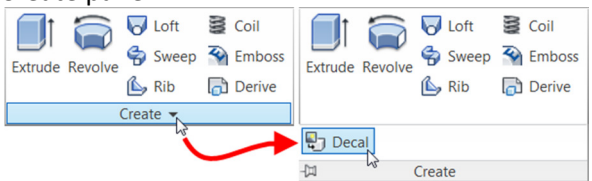
- g) Next center the image within the sketch by apply a horizontal constraint between the midpoints labeled (1) and a vertical constraint

between the midpoints labeled (2).



h) Finish the sketch

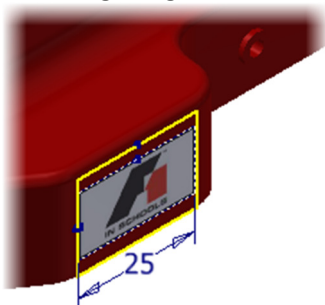
- 3) The last step is to create the decal with the Decal command. To start the Decal command click the down arrow on the bottom of the 3D Model tab > Create panel.



- a) Next select the image that will be used to create the decal.  
 b) In the Decal dialog box click the Face button will automatically become active, select the face on the sketch that the image was placed on.

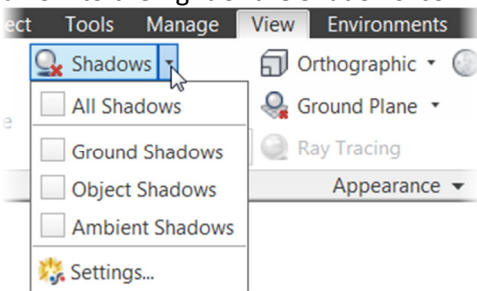


- c) Click OK to create the decal.  
 d) If desired repeat these steps to create the F1 in Schools decal using the file *F1 in Schools Logo.png* with a width of **25 mm** and centered on the sides of the car, shown in the following image.



- e) To see the completed car, open the file *Ex4 F1 Car Body Appearance Completed.ipt*.  
 4) Close all Inventor files.  
 5) Next you enhance the appearance and clarity of the car assembly by applying shadows. Start by opening the file *Ex4 F1 Race Car Assembly - Appearance.iam*.

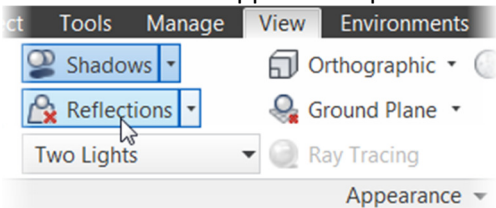
- 6) When applying shadows there are four options to choose from.
- All Shadows:** All three shadow effects are made active and display.
  - Ground Shadows:** Casts a model shadow onto the ground plane. This effect does not require the ground plane to be visible.
  - Object Shadows:** Casts, and receives model shadows based on the position of the active lighting style.
  - Ambient Shadows:** Casts shadows in corners and cavities to visually enhance the transition of shape changes.
- 7) Practice applying the different shadows by clicking the shadow option on the View tab > Appearance panel > and click the drop down arrow to the right of the Shadows icon.



- When done check the All Shadows option.  
**Note: After a shadow option is selected, it can be turned on and off by clicking the top level Shadows icon.**



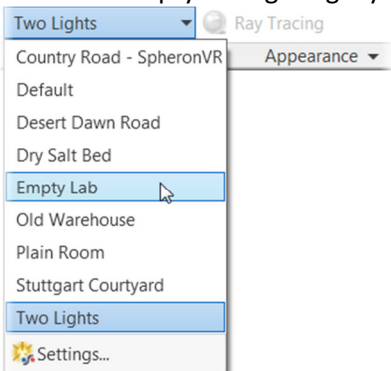
- 8) Next turn on reflections by clicking the Reflections on the View tab > Appearance panel.



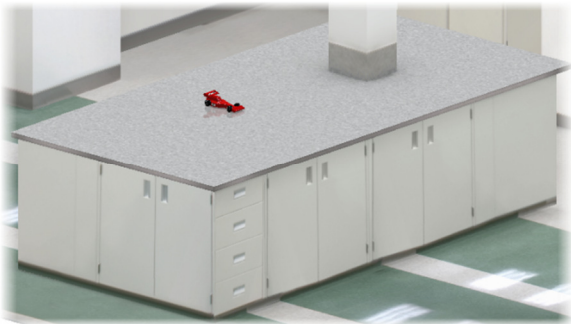




- 9) You can also change the background and lighting options by selecting from a list of preset light styles. From the View tab > Appearance panel select an option from the list of lighting style. The lighting styles that have a background are 3D, which means as you change the viewpoint you will see different aspects of the background. Make the Empty Lab lighting style current.

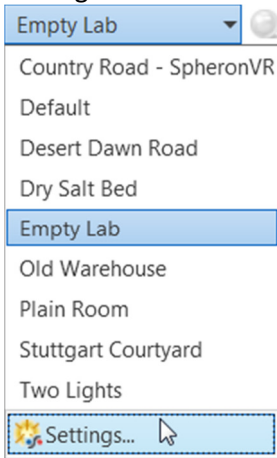


- a) Change the viewpoint so your screen resembles the following image.

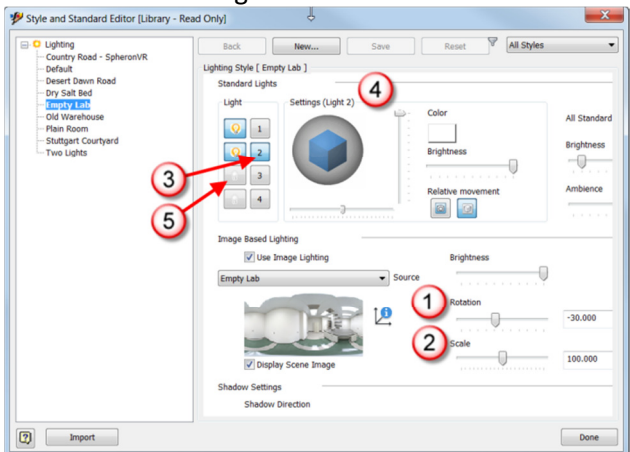


- b) Next you adjust the background. Click the Empty Lab option on the View tab > Appearance panel and from the list click

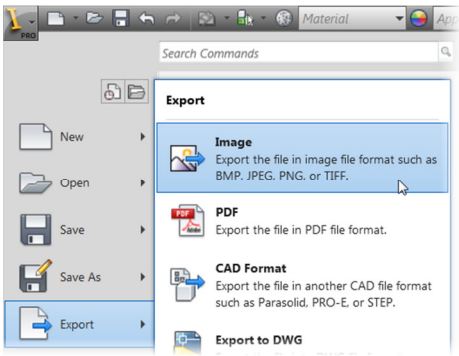
## Settings.



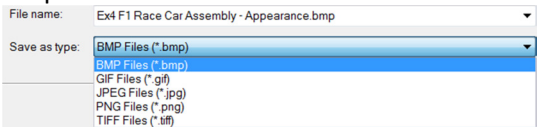
- c) Next you rotate the image by clicking and dragging the Rotation slider, labeled (1).
- d) Practice scaling the image by clicking and dragging the Scale slider, labeled (2).
- e) Next you adjust the lights. Make Light 2 current by clicking the 2 button, labeled (3).
- f) In the Settings (Light 2) area, labeled (4), practice changing the angle of the lights by clicking and dragging on the horizon and vertical sliders.
- g) In the same Settings (Light 2) area experiment changing the color and brightness of the light.
- h) If desired turn on additional lights by clicking the light bulb in the Lights area, labeled (5).
- i) When done editing the background, click Save in the dialog box.



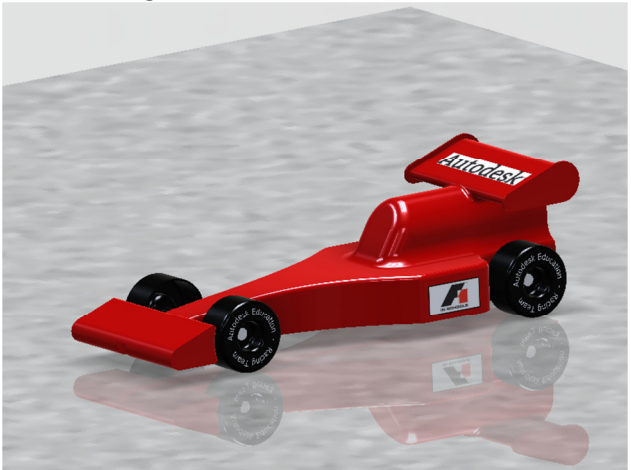
- 10) Change the viewpoint as desired.
- 11) If desired, practice applying other backgrounds.
- 12) Next you save an image of what is currently seen in the graphics window. Use the navigations commands to change the viewpoint, zoom and pan as need.
  - a) To create an image file, in the upper left corner of the screen click the Application Menu > Export > Image.



- b) If desired change the file type to be created by clicking on the default file type in the Save as type area and select the file type from the drop list.



- c) Click the Save button and open the image file. Following is an example of the exported image.



- 13) Save and then close the file.  
14) Close Inventor.