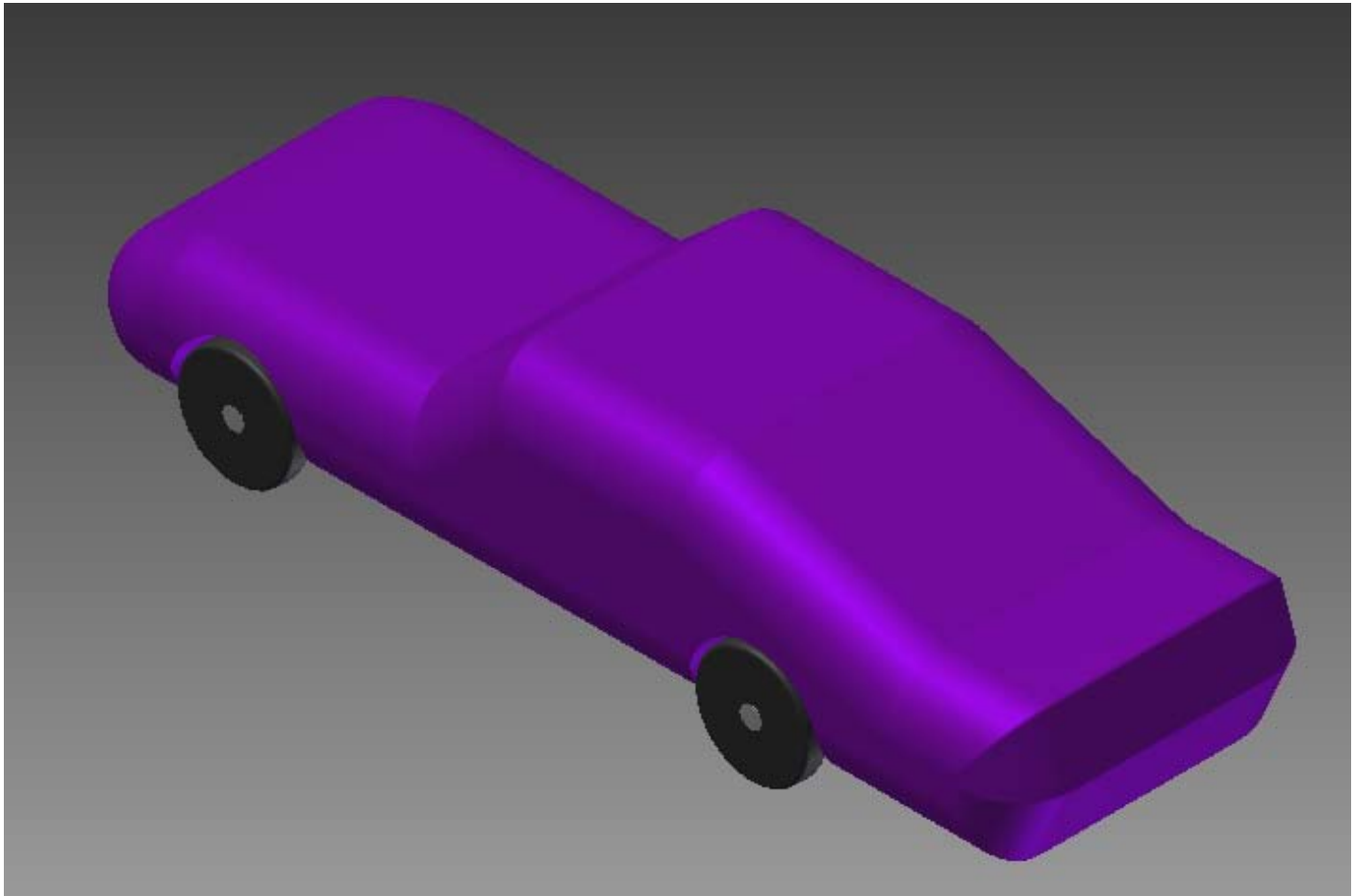
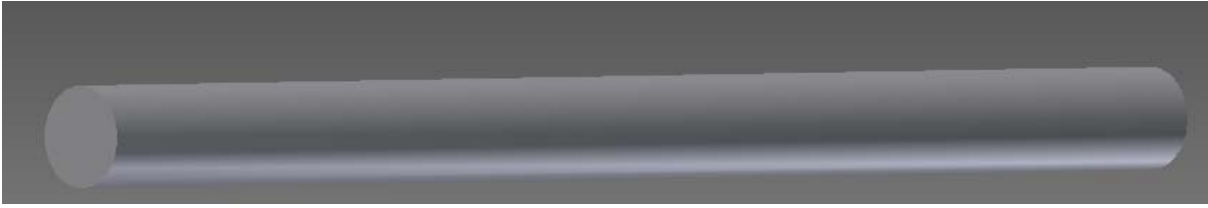


Vehicle Design – Inventor Professional

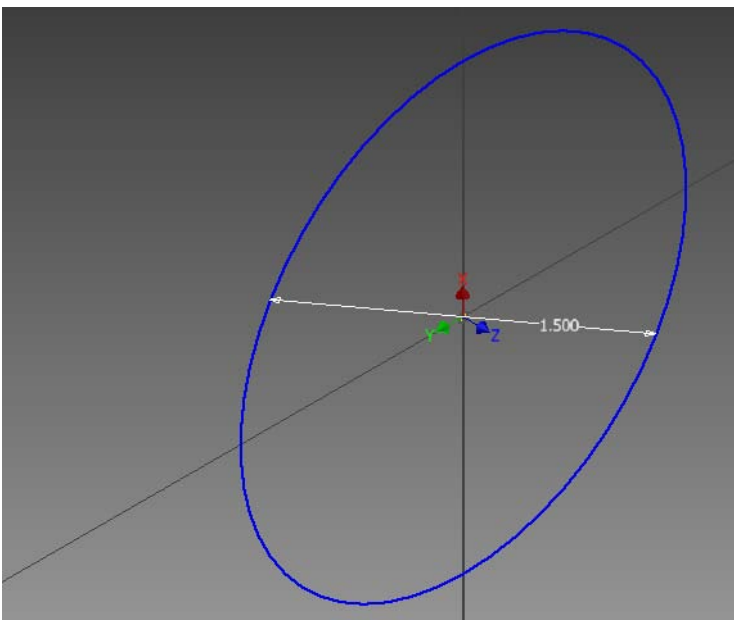


In this tutorial, you will construct a *vehicle design*. Your vehicle will have a body, four wheels and two axles. You will learn *advanced sketching techniques* and *assembly procedures*.

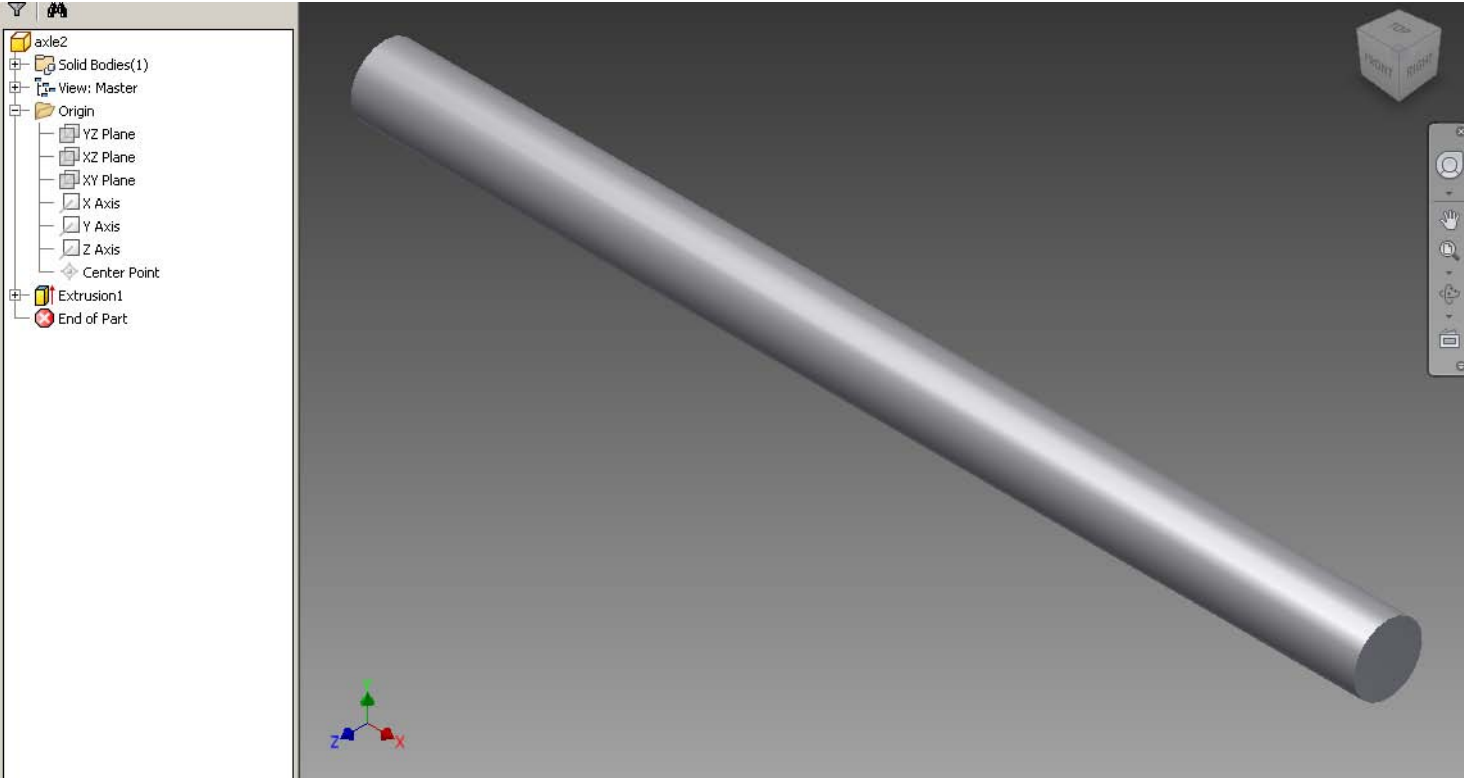
Part I – Create the Axle



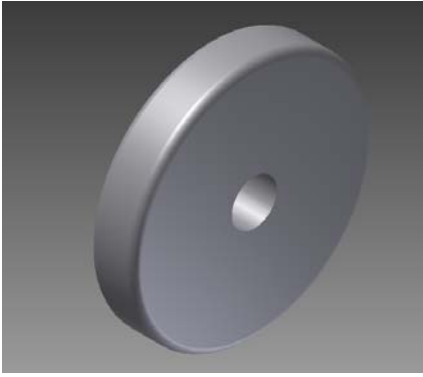
1. Open Inventor. In the *Get Started* tab, click on **Projects > New > New Single User Project**. Click *Next*.
2. In the *Project Name* box, type **Vehicle Design**. In the *Project (Workspace) Folder*, Click on the browser box to select your **H:\ drive**.
3. In the *Browse for Folder* box, navigate to your **Intro to CAD** folder. Click on *Make New Folder*, and name the new folder **Vehicle Design**. Click *OK > Finish > Apply > Done*.
4. *You will use this process whenever you start a new project.*
5. In the *Get Started* tab, click on **New**. In the *New File* box, click “**standard.ipt**”, then **OK**. This will open a new, standard part file in Inventor.
6. Save the new part as “**axle_INL_p5**”, where “**INL**” should be **YOUR** initials and “**p5**” should be **YOUR** class period. Since you set up a Project Folder, your new part will automatically be saved there.
7. Go to **Tools > Document Settings > Units**. Set the *Length* units to **Inches**. Leave the rest of the settings unchanged.
8. Create a *New Sketch* on the “**YZ**” plane.
9. Sketch a circle of 1.5”



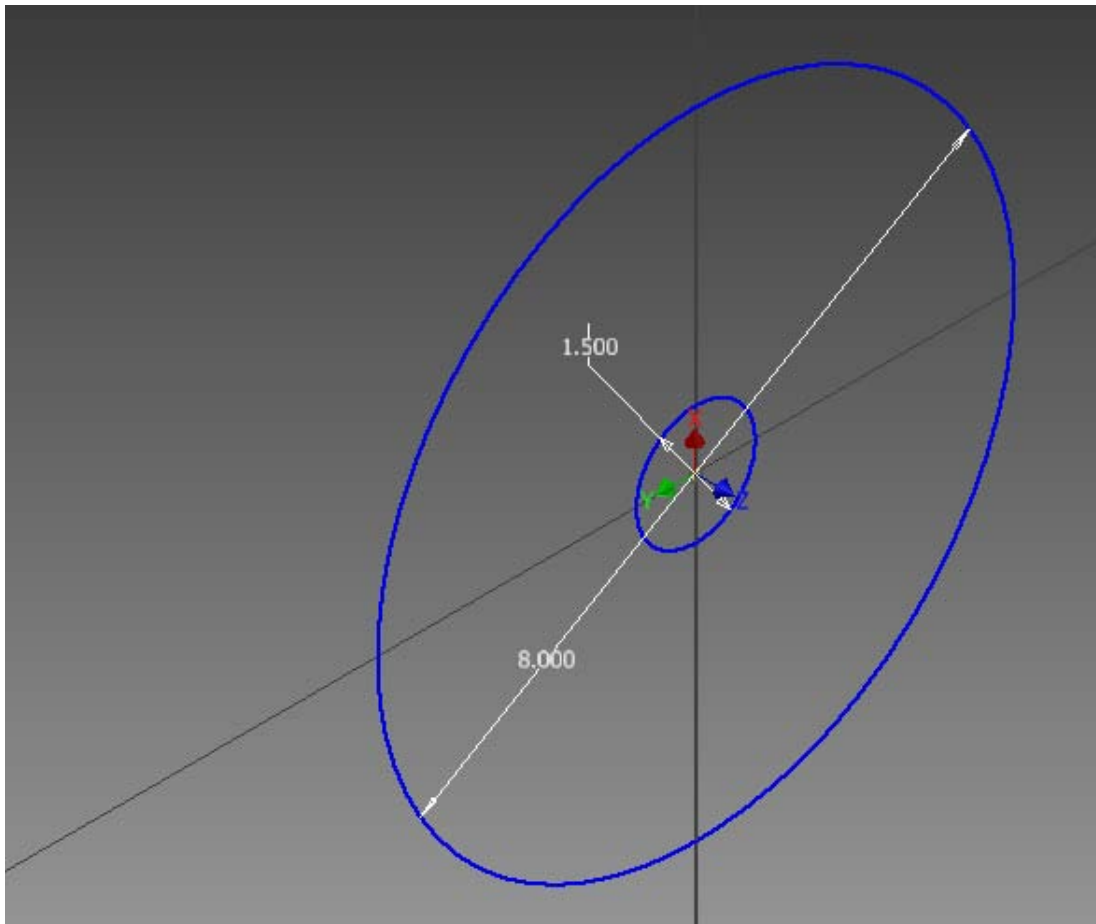
10. Extrude the circular profile to 22 inches. **SAVE YOUR WORK** [Watch Video 1](#)



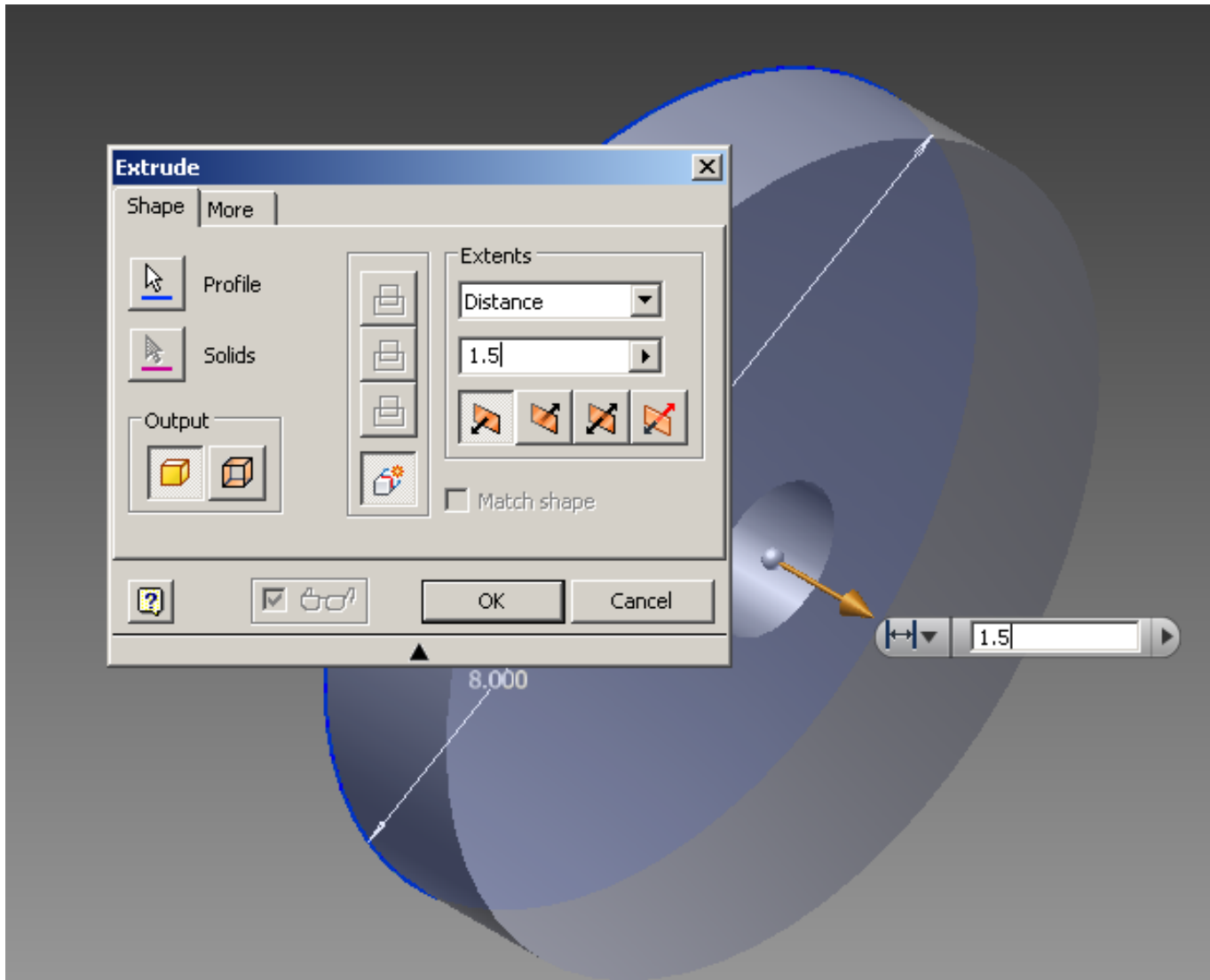
Part II – Create the Wheel



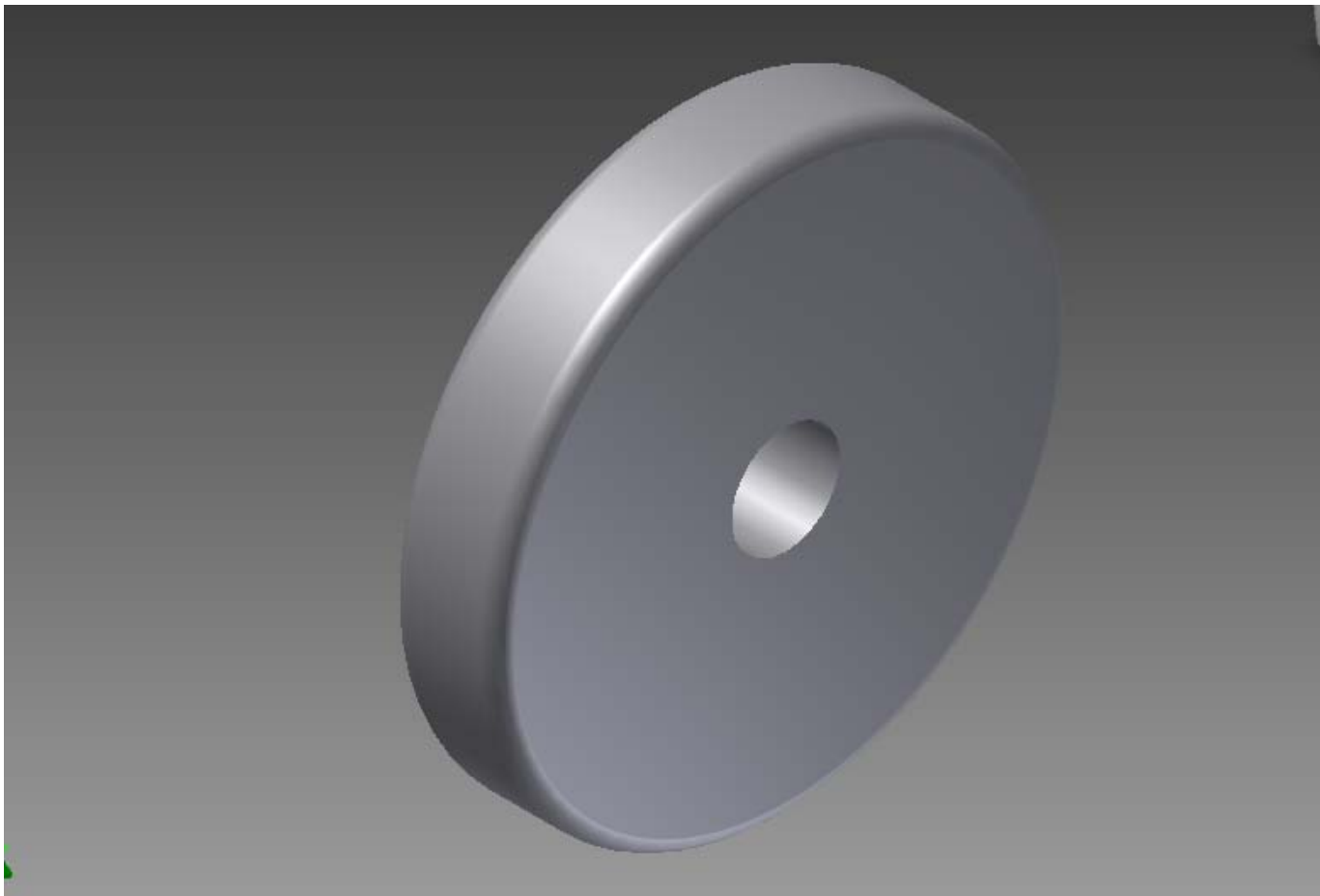
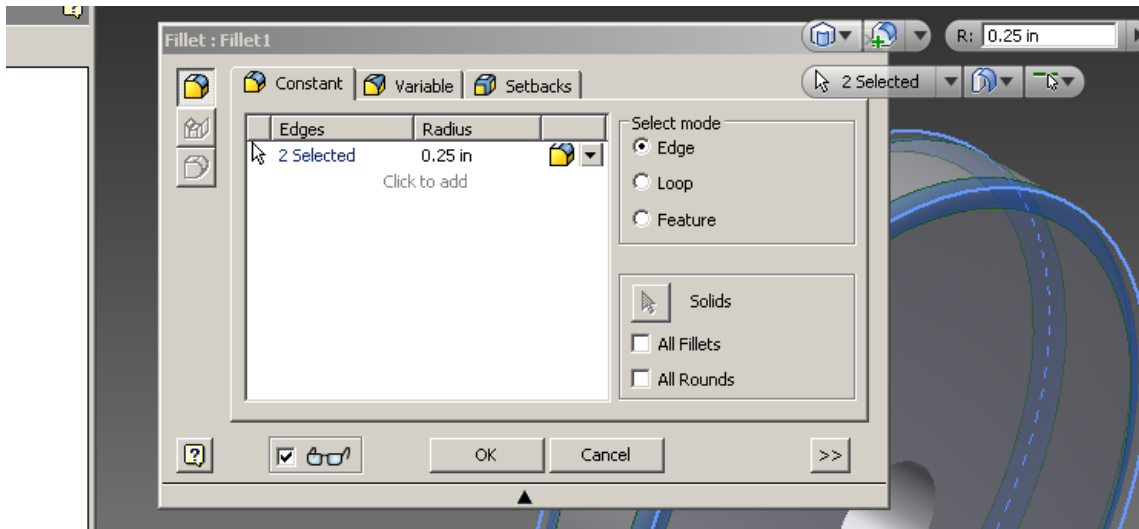
1. In the *Get Started* tab, click on **New**. In the *New File* box, click “**standard.ipt**”, then **OK**. This will open a new, standard part file in Inventor.
2. Save the new part as “*wheel_INL_p5*”, where “INL” should be *YOUR* initials and “p5” should be *YOUR* class period. Since you set up a Project Folder, your new part will automatically be saved there.
3. Go to **Tools > Document Settings > Units**. Set the *Length* units to *Inches*. Leave the rest of the settings unchanged.
4. Create a *New Sketch* on the “YZ” plane.
5. Sketch concentric circles of **1.5** inches and **8** inches.



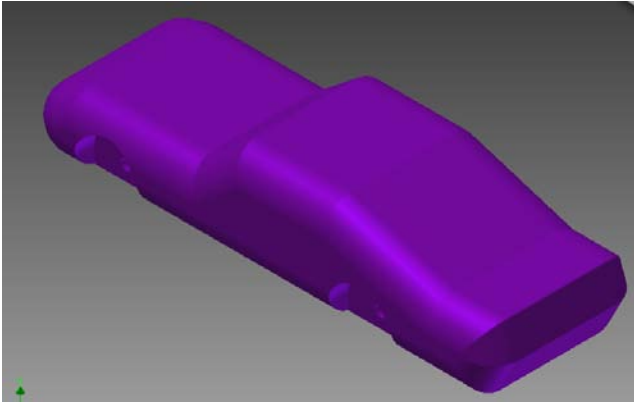
6. Extrude the profile to **1.5** inches.




7. Use the **Fillet** tool  to create a .25 inch **fillet** on the outside edges of the wheel. **SAVE YOUR WORK** [Watch Video 2](#)

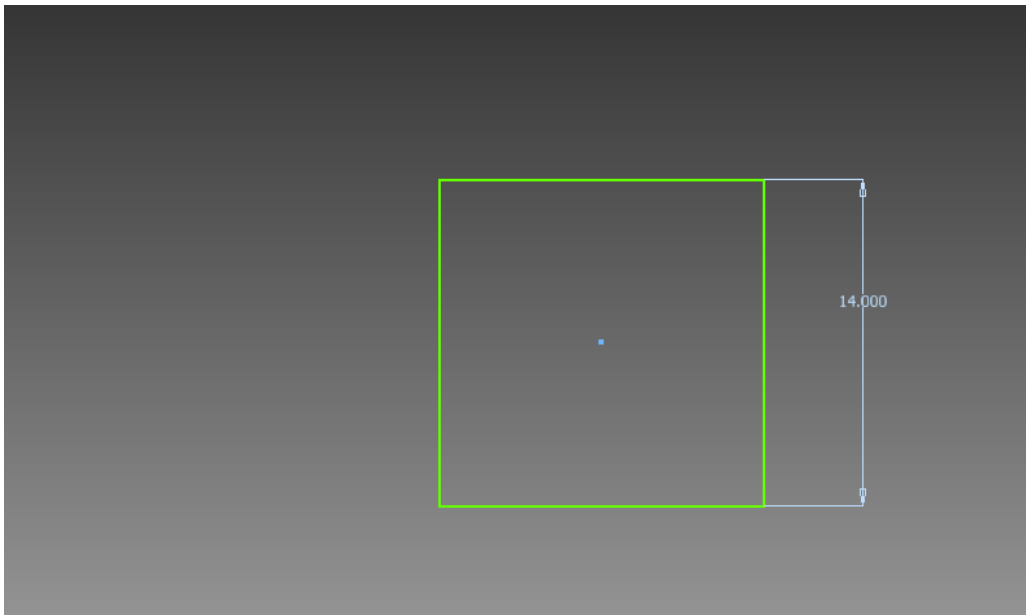


Part III – Create the Vehicle Body

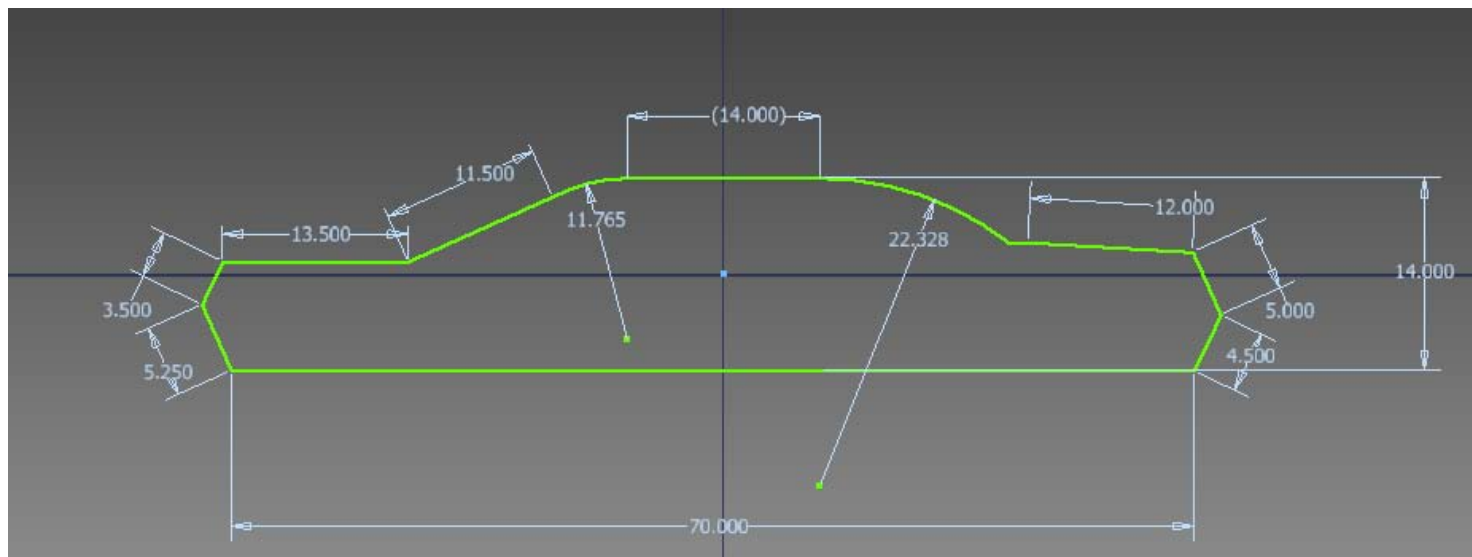



1. In the *Get Started* tab, click on **New**. In the *New File* box, click “**standard.ipt**”, then **OK**. This will open a new, standard part file in Inventor.
2. Save the new part as “*body_INL_p5*”, where “INL” should be *YOUR* initials and “p5” should be *YOUR* class period. Since you set up a Project Folder, your new part will automatically be saved there.
3. Go to **Tools > Document Settings > Units**. Set the *Length* units to *Inches*. Leave the rest of the settings unchanged.
4. Create a *New Sketch* on the “XY” plane.
5. Click on the **Polygon** tool  *Polygon* . Create a *4 sided polygon*, starting at the **Origin Point**. By using the Polygon tool, instead of the rectangle tool, you will create a four-sided, equilateral polygon (a square). This allows you to dimension just one side, as well as keep your design symmetrical for later redesigns.
6. Be sure to snap the square so that the sides are parallel to the “X” and “Y” axes.
7. Dimension one of the sides of the square. Set the value at *14*”. Click on *Finish Sketch*.

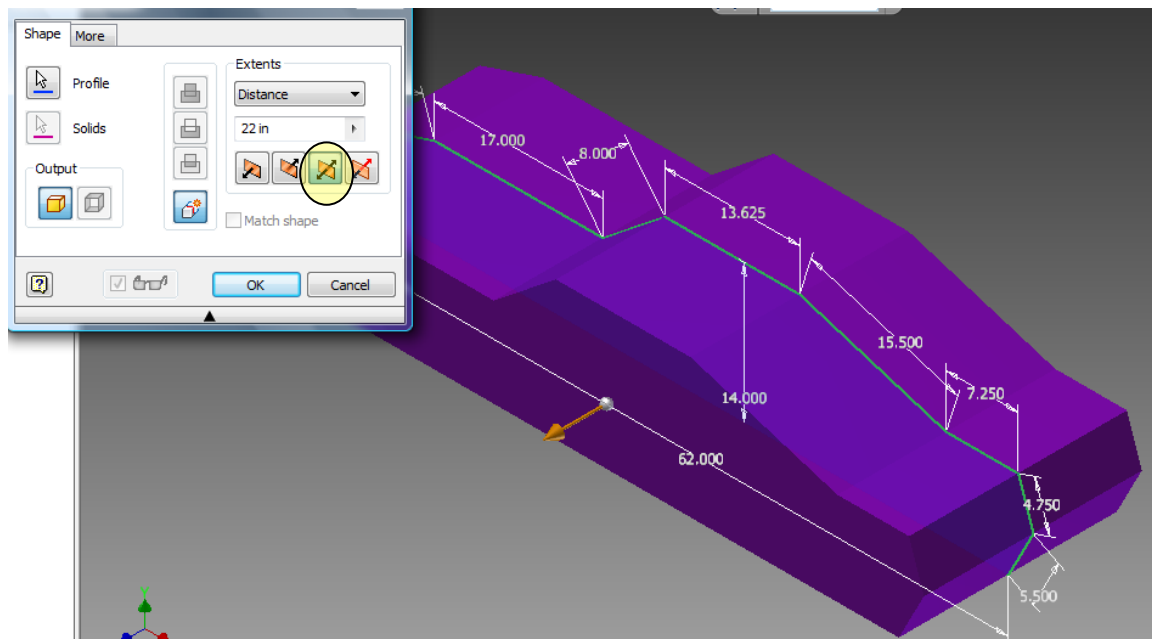
[Watch Video 3](#)

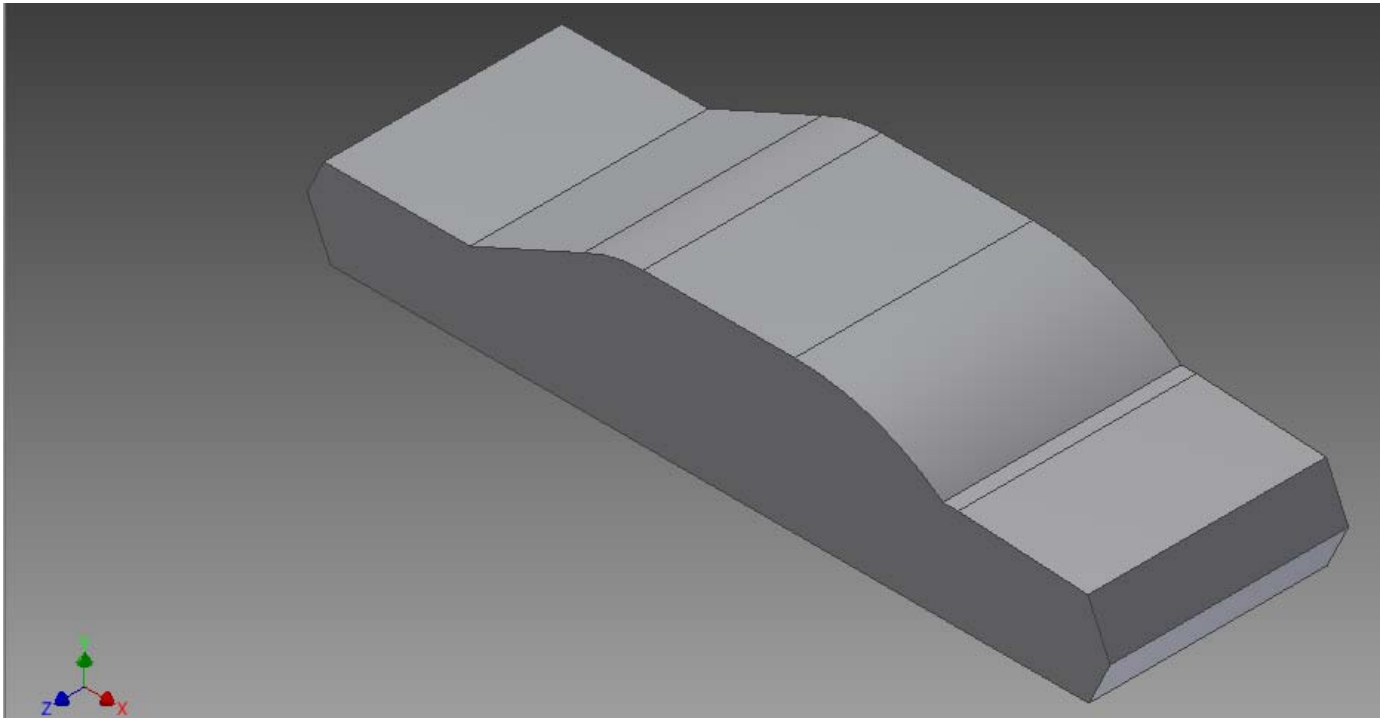


8. **Click**, pause, then **click again** on the sketch in the **Browser** window. **Rename** the sketch to “**profile**”.
9. Using the line tool and the arc tool, sketch **a design of a profile of a vehicle**. Be sure to include ALL the needed dimensions. Your design should be unique. THE DESIGN INCLUDED IN THIS TUTORIAL IS AN **EXAMPLE** OF A DESIGN, BUT YOUR DESIGN SHOULD BE SIGNIFICANTLY DIFFERENT.

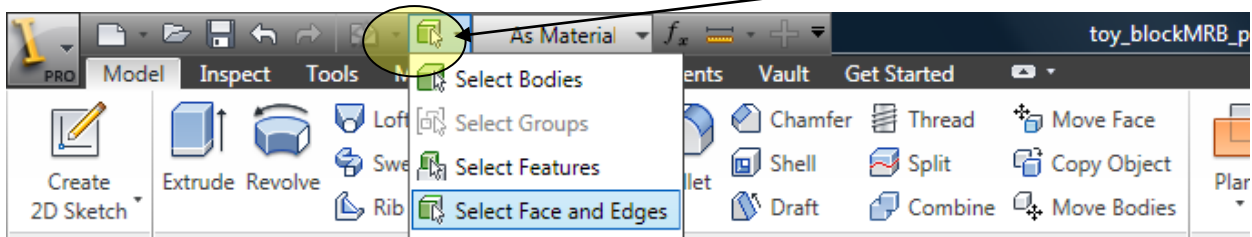


10. Click on the Extrude icon . Select the profile, and extrude the profile to 22”. Rename the extrusion to **extBODY**. See the figure below. **SAVE YOUR WORK** [Watch Video 4](#)

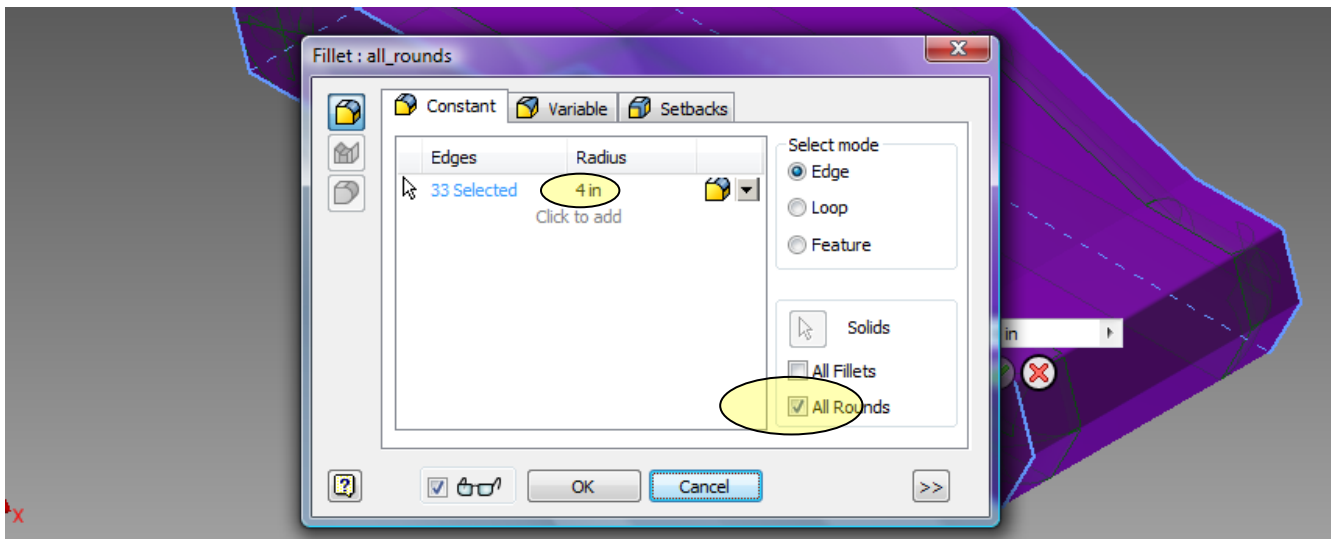




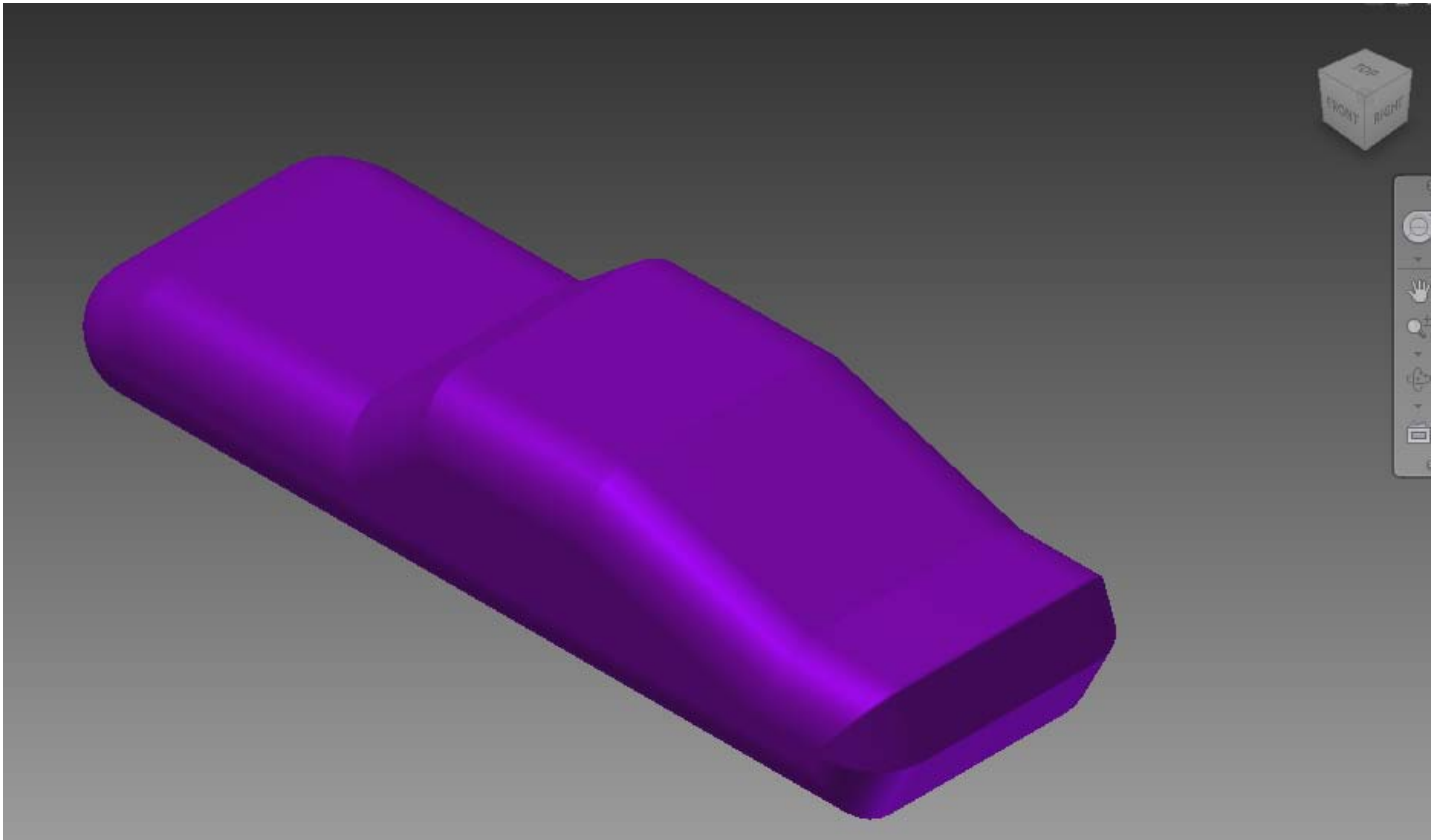
11. Be sure *Select Faces and Edges* is selected in the Select drop down menu. The **Select** menu is located in the **QAT** – just above the *ribbon*.



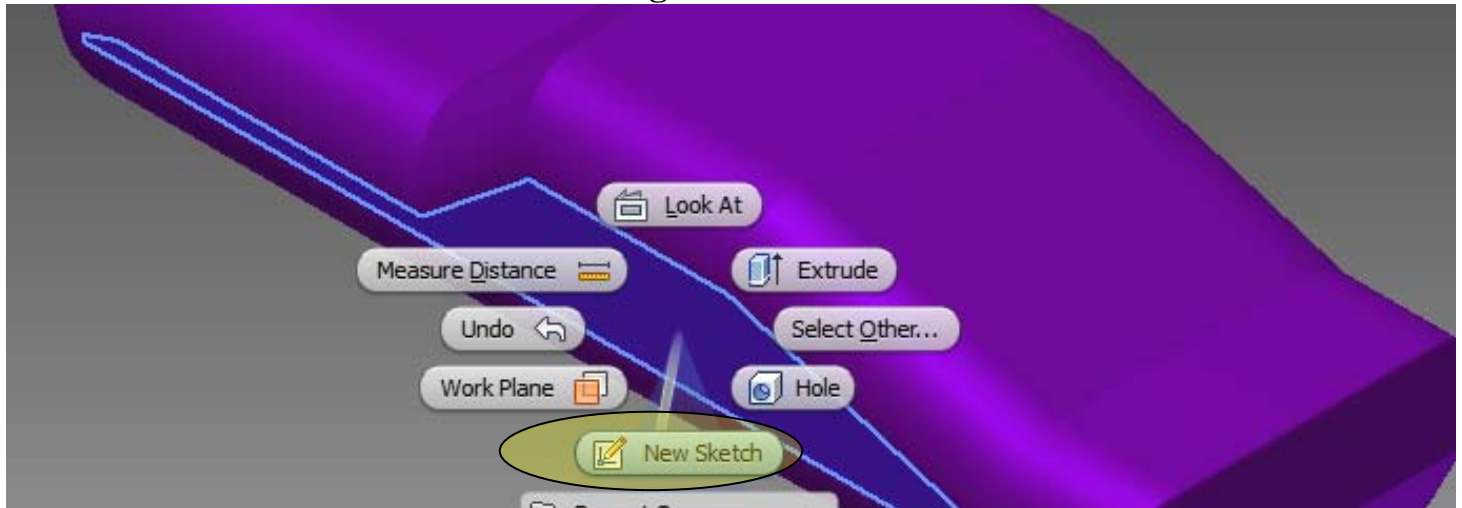
12. Select the *Fillet tool* . Check off **All Rounds** and change the *radius* to to a number between 2 and 5 inches. See below.



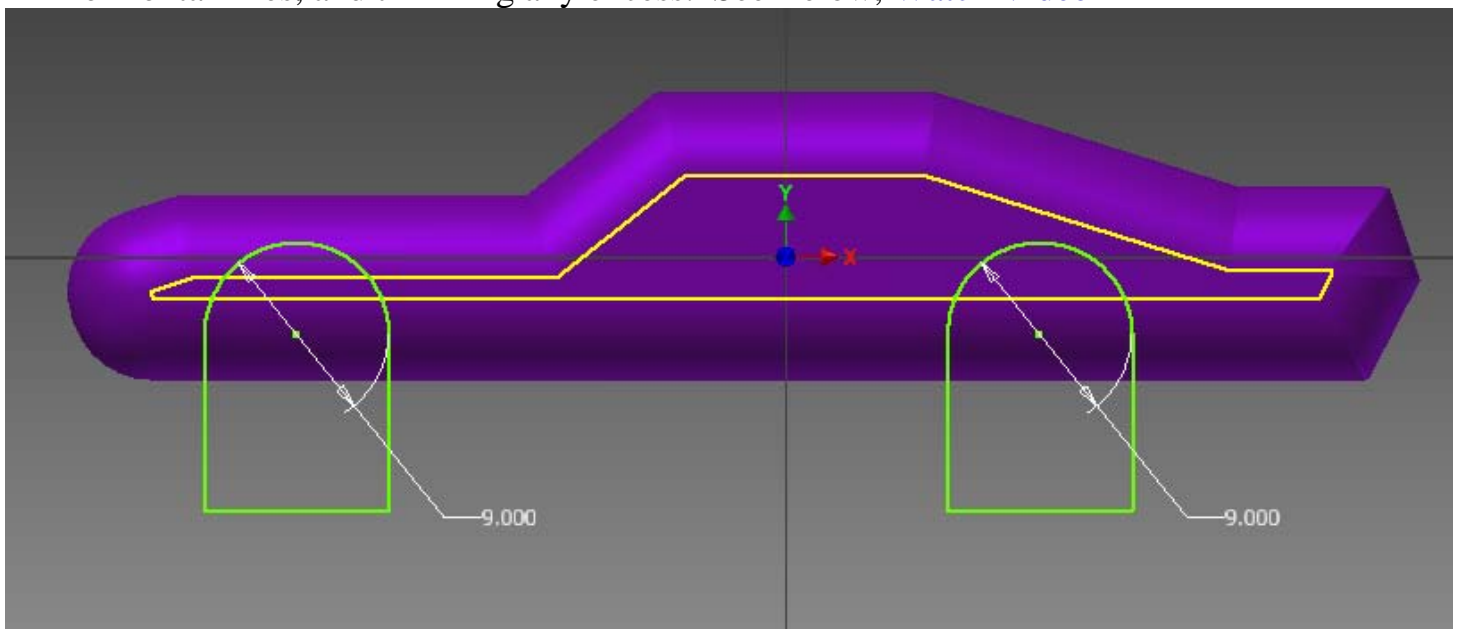
13. Notice also that by selecting *All Rounds*, you have selected *all of the edges of the vehicle*. You may want to hold **CTRL+CLICK** to *deselect* some of the edges that you don't want rounded, or that you want to round at a different radius later. Click **Apply**, then **Cancel** or *close the box*. Your vehicle should now be rounded like the one below. REMEMBER – YOU MAY HAVE TO ROUND SOME INDIVIDUAL EDGES, DEPENDING ON YOUR DESIGN INTENT. **SAVE YOUR WORK** [Watch Video 3](#)



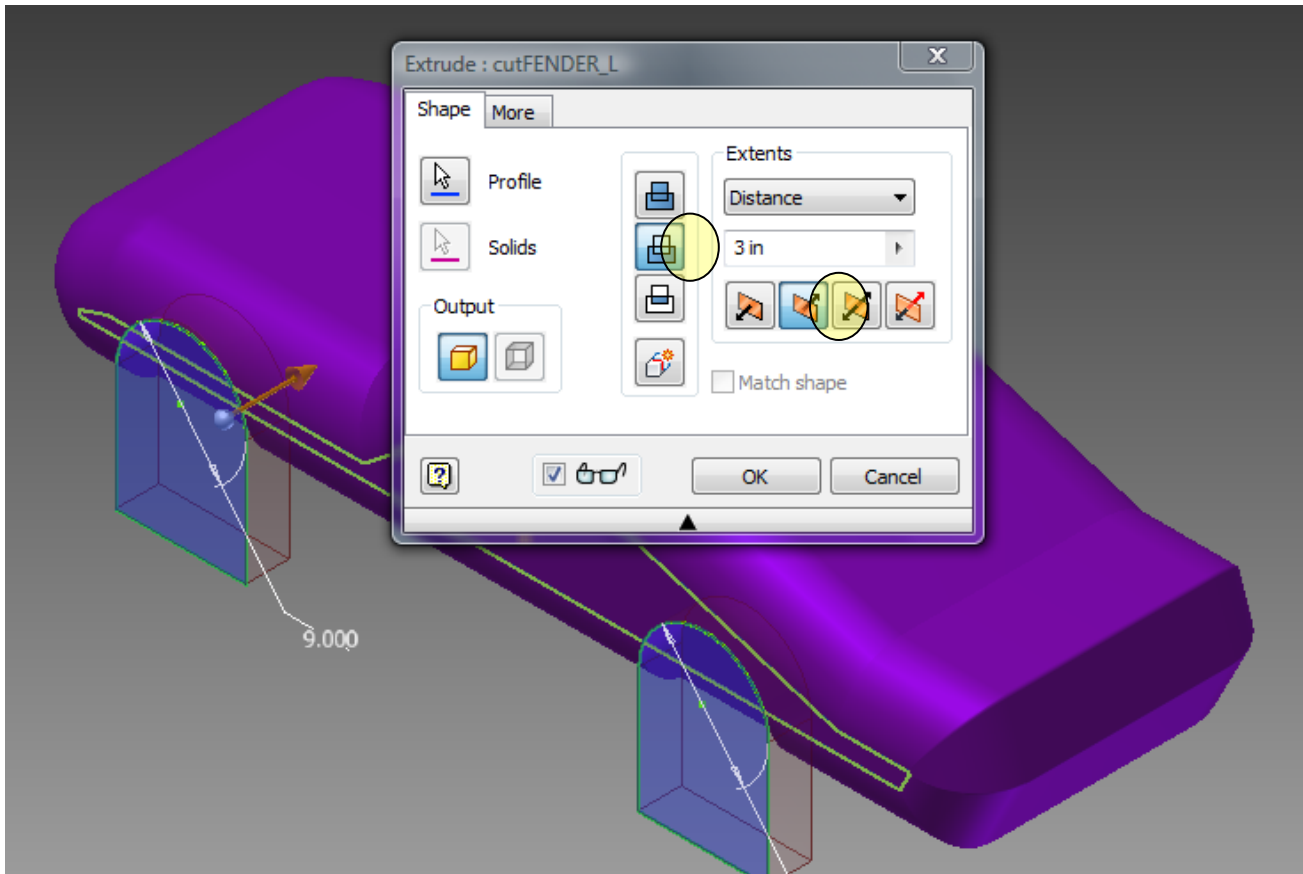
14. Select the side face of the vehicle > **Right Click** > *New Sketch*.




15. Create the *fender cuts* like the ones below. You will sketch two collinear circles, then sketch tangent vertical lines. You will finish the sketch by enclosing the profile with horizontal lines, and trimming any excess. See Below, [Watch Video 4](#)



16. **Extrude** the profile > **remove** material > **below** the workplane. See below:

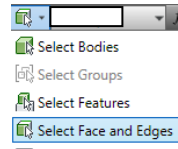


17. Be sure to rename the extrusion to **cutFENDER_L**.

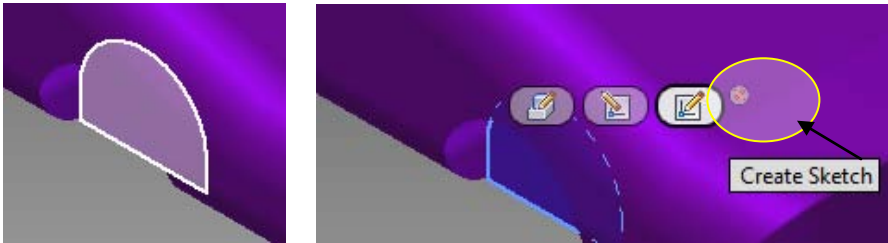
18. Next, you will **mirror** the fender cuts so that they appear on the opposite side of the vehicle. Select the **Mirror** tool  from the **Pattern** panel of the Model tab.

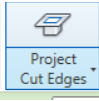
19. A box will appear. You will need to select a **feature** and a **mirror** plane. Select the **cutFENDER_L** as the feature, and the XY plane as the **mirror** plane. Click **OK**.

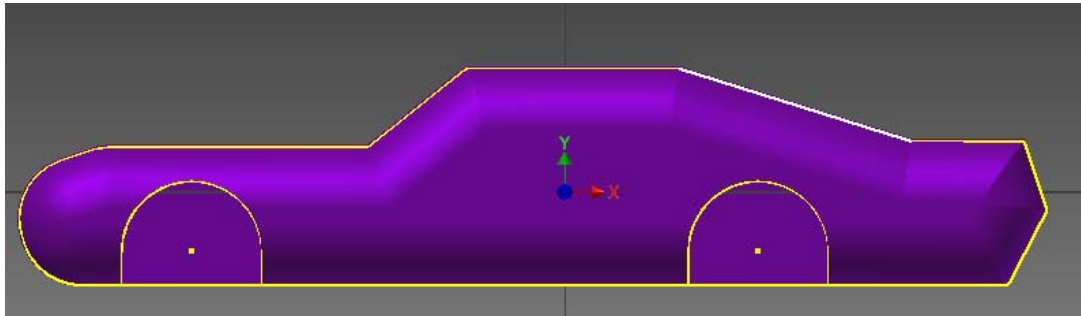
20. Next, you will create the axle holes that will hold the axle and wheels. Be sure Select



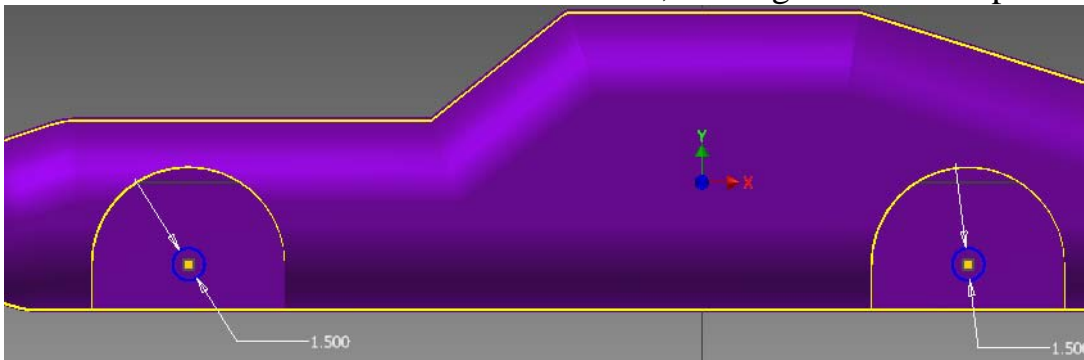
Faces and Edges is active from the **Selection** menu on the Quick Access Toolbar (QAT). **Click** on the inside face of the fender cut (below) >**Create Sketch**.



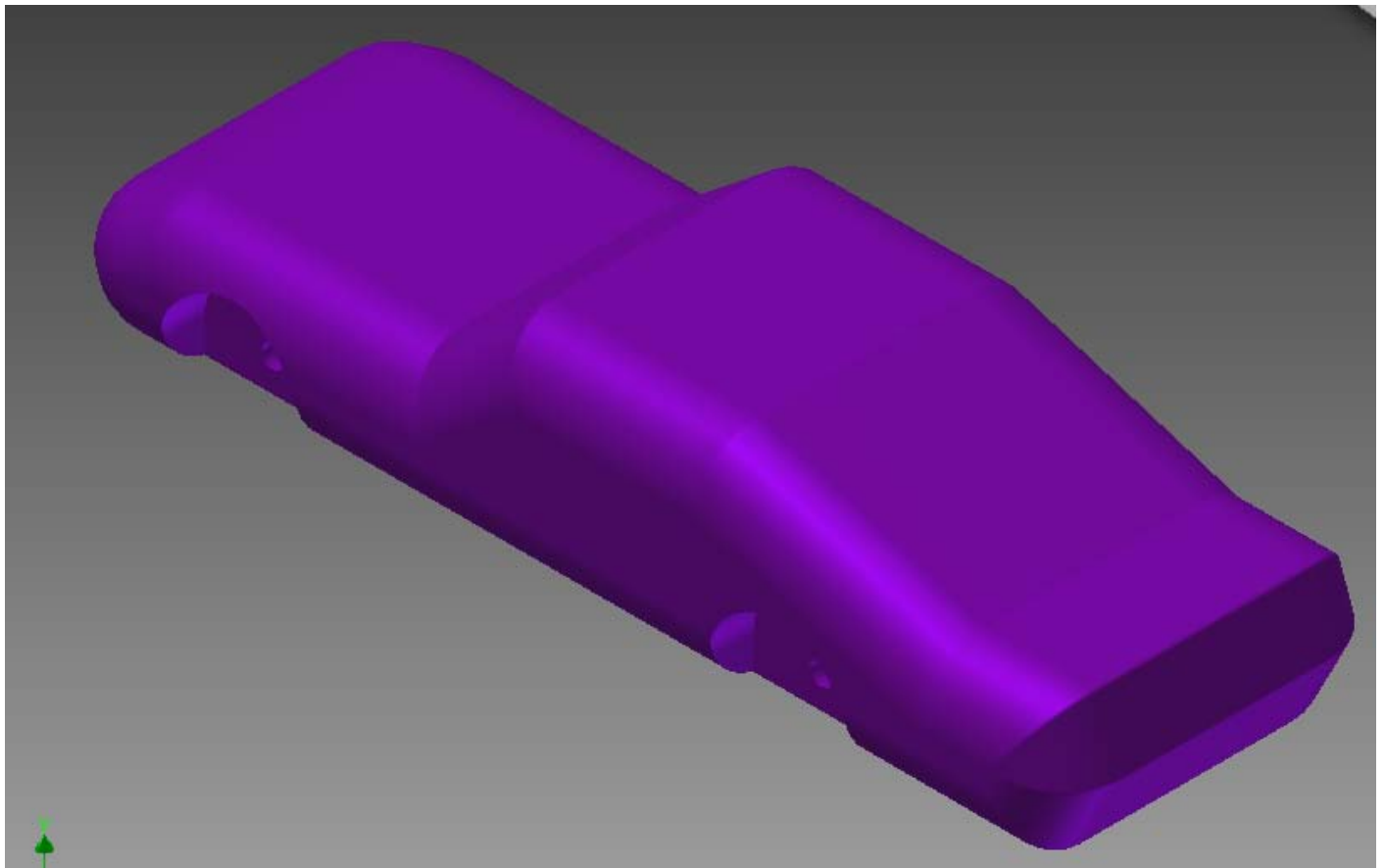
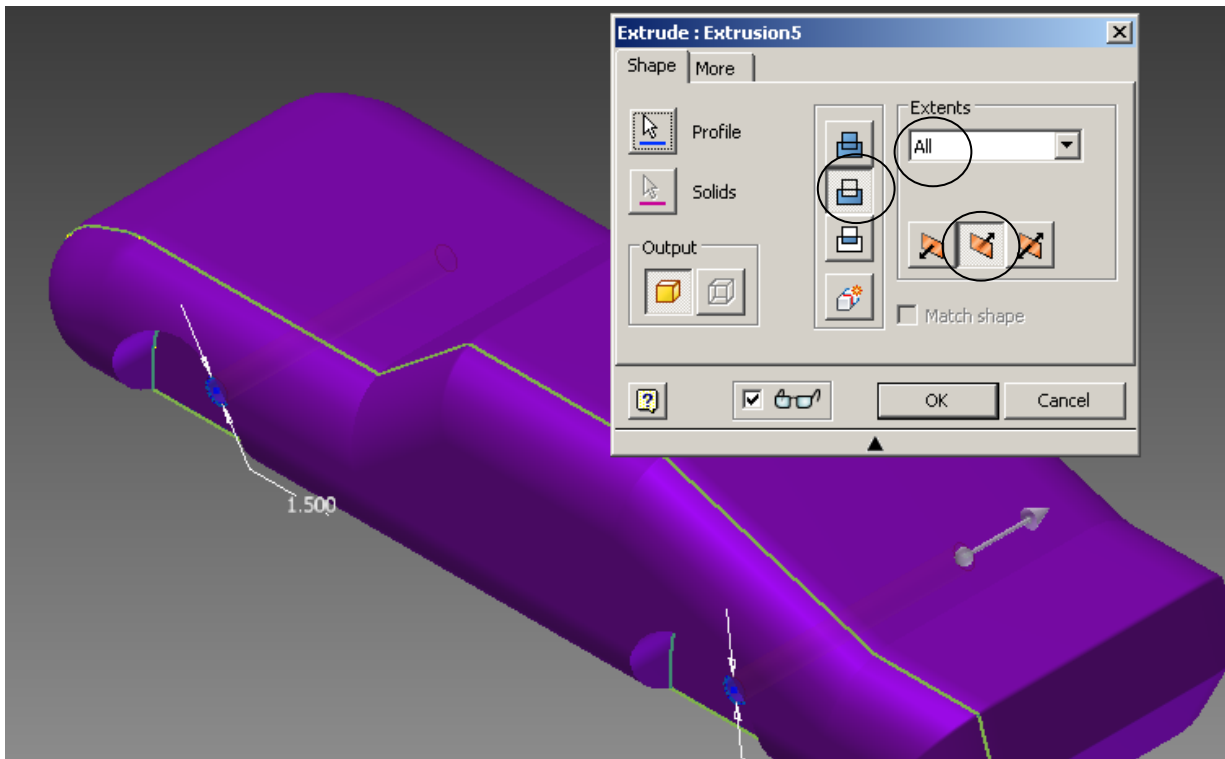
21. Select ***Project Cut Edges***  from the **Draw** panel. This will create sketch lines from the edges of the previous extrusion and cut.



22. Sketch ***concentric*** circles of 1.5 inches, starting at the center point of the fender arcs.

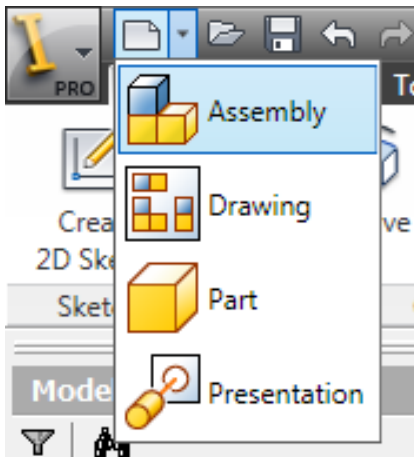


23. Extrude the holes as a ***Cut > Below Workplane > through All***. **SAVE YOUR WORK** [Watch Video 5](#)



Part IV – Assemble the Vehicle

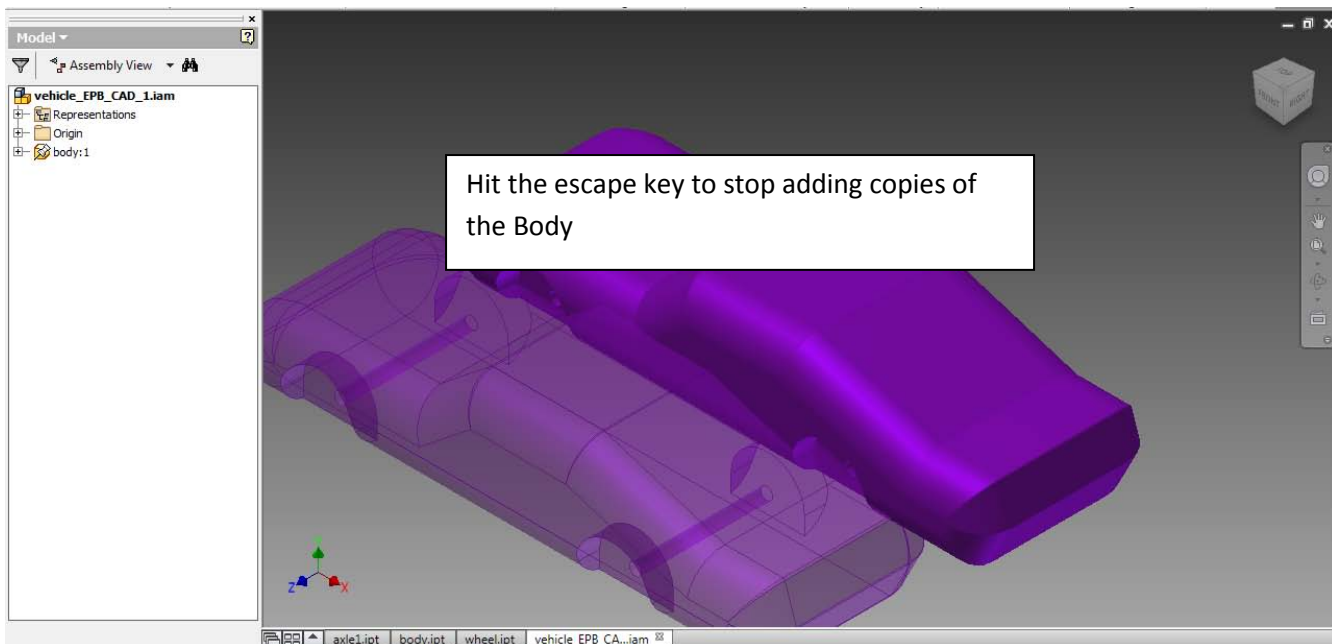
1. Click on *New > Assembly*



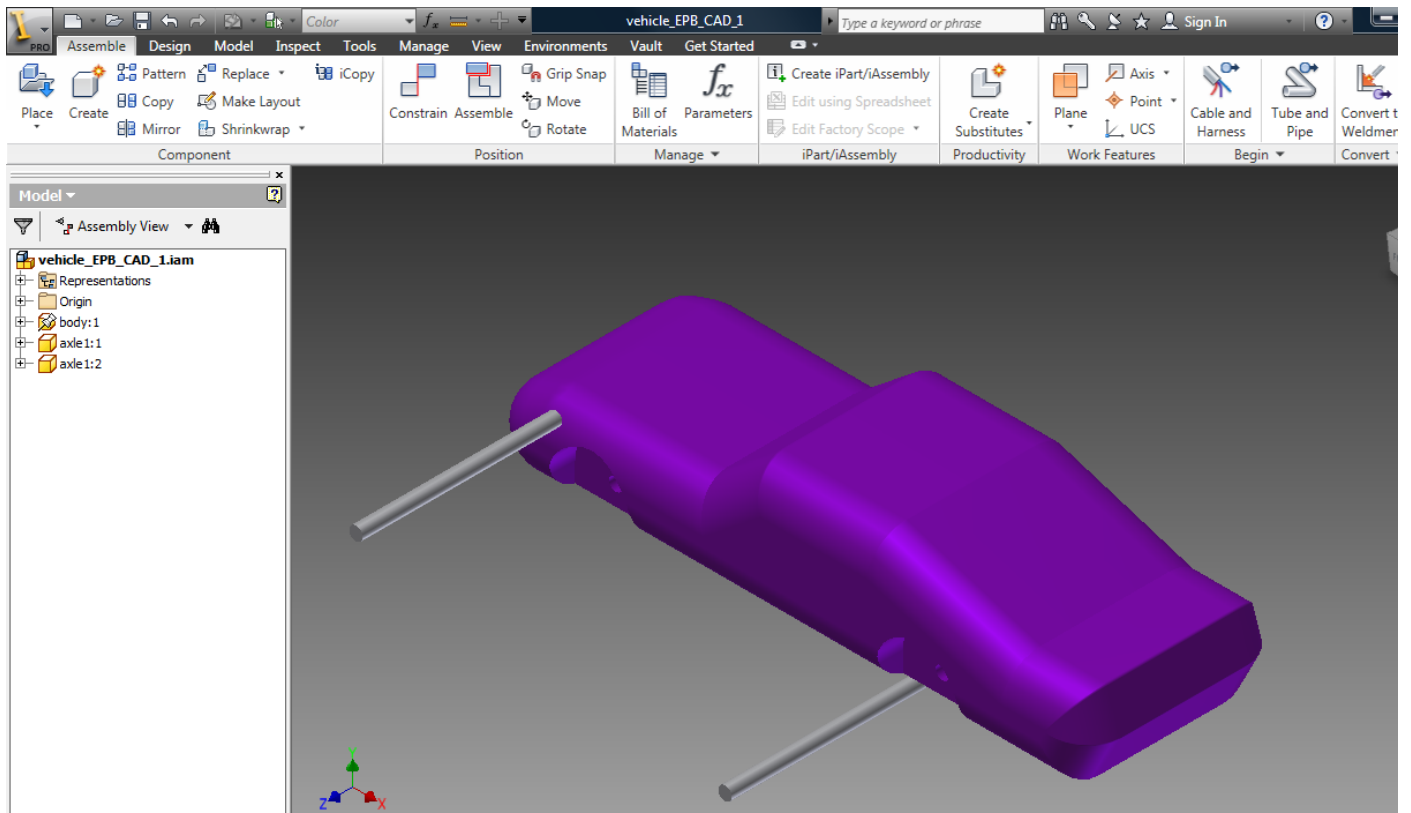
2. Save As..*vehicle_INL_CAD_5* using your initials and your class period.
3. Click on **Assemble > Component > Place**.



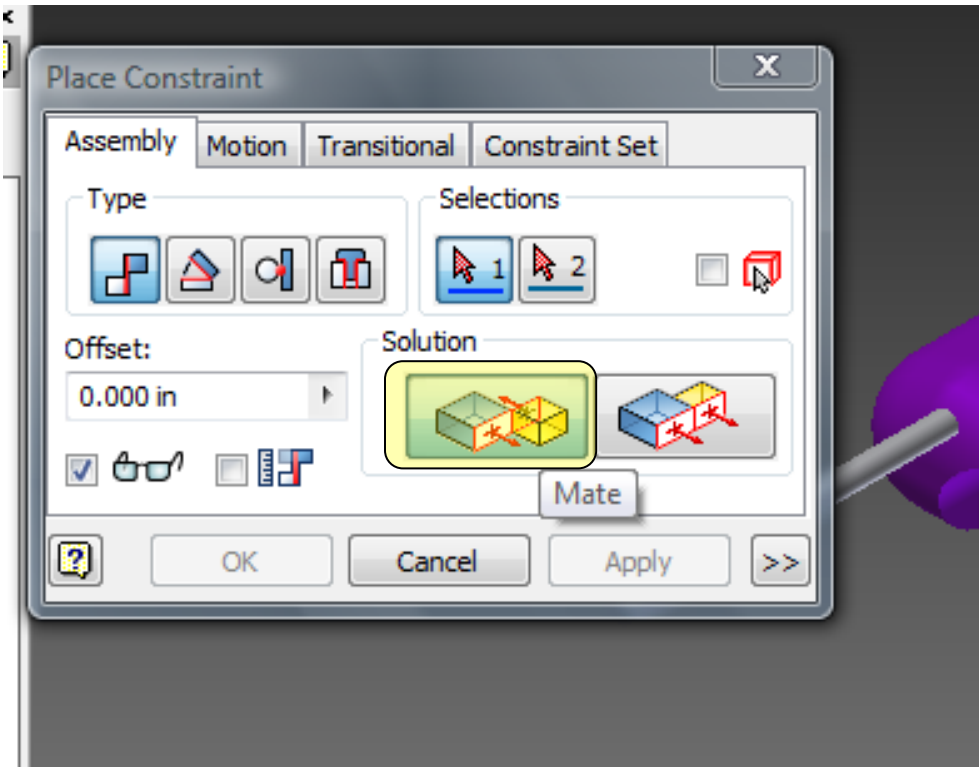
3. Browse for the Body of your vehicle. Inventor will place one copy of the Body in the *graphics window*. Hit the escape key to cancel adding additional copies of the Body.



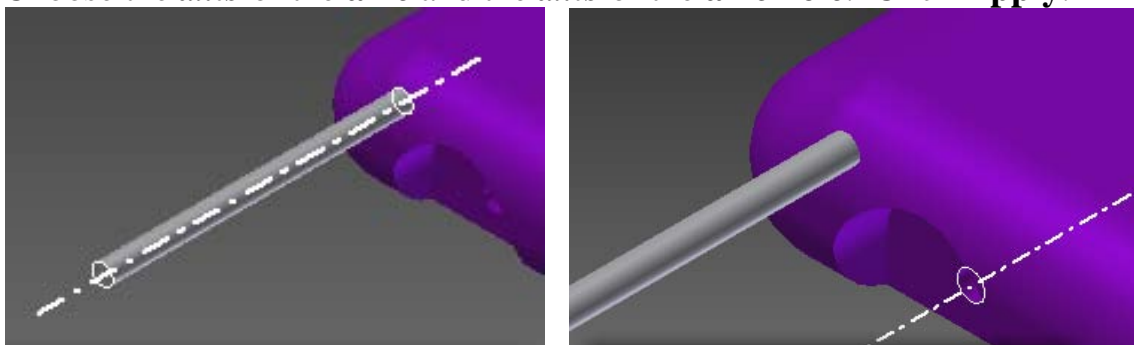
4. Place two axles onto the *Graphics Window*.



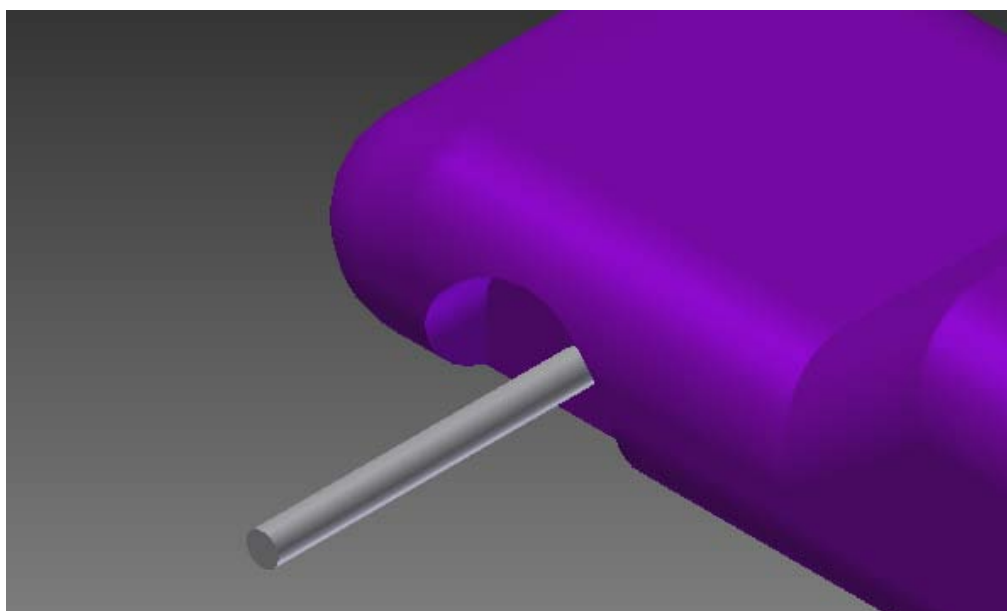
5. Click on the **Assemble > Position > Constrain**. Choose *Mate* from the *Solution* area.



6. Choose the *axis* of the **axle** and the *axis* of the **axle hole**. Click **Apply**.

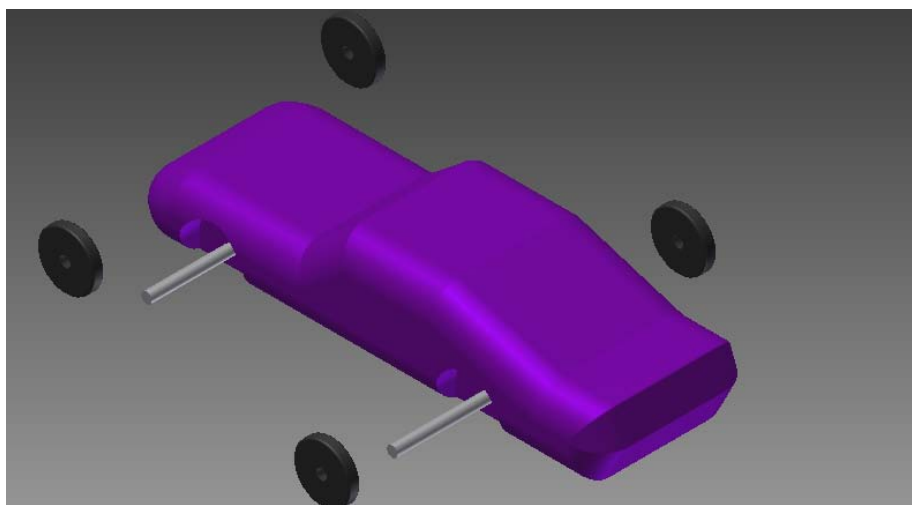


The axle should now be concentric with the axle hole.

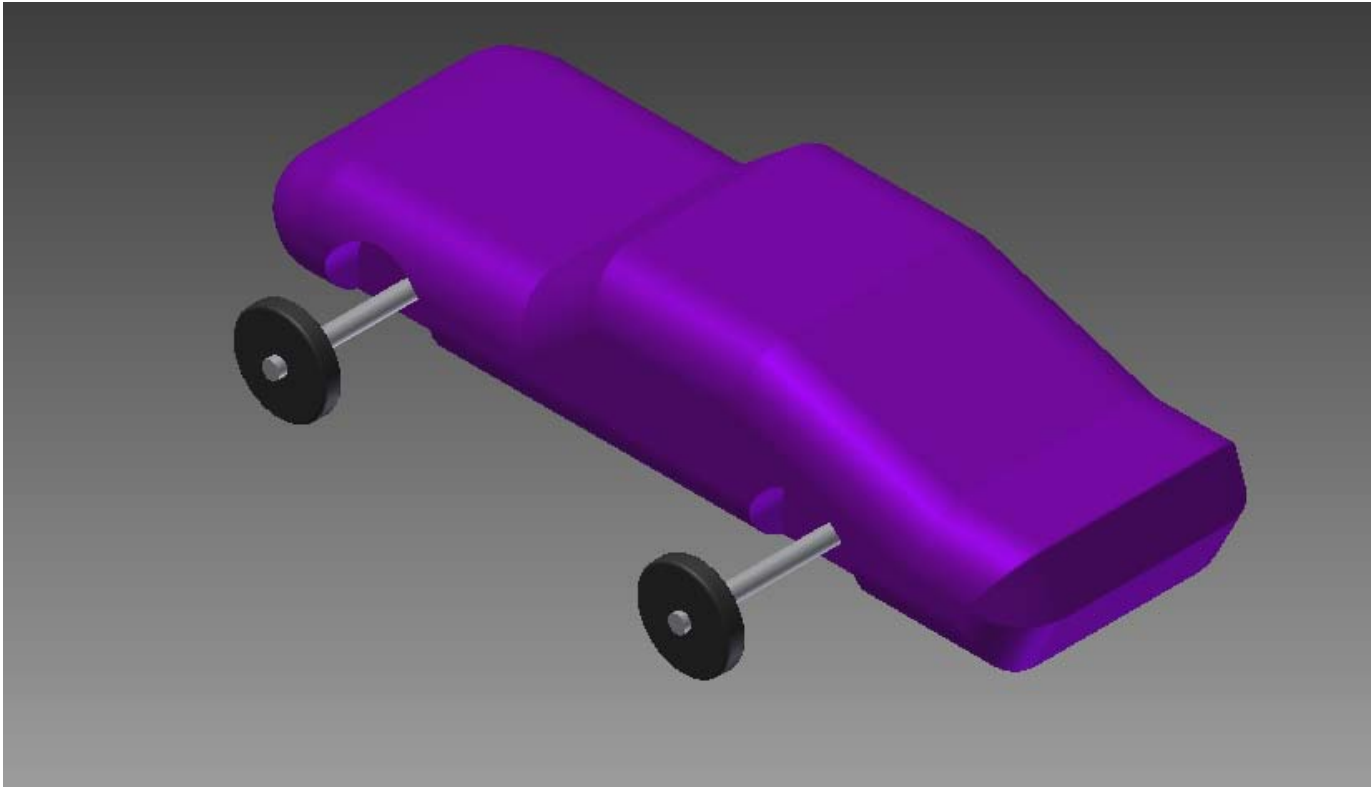


7. With the **Constrain** window still active, repeat the procedure for the other axle and axle hole. Click **Apply** > **Cancel** when finished.

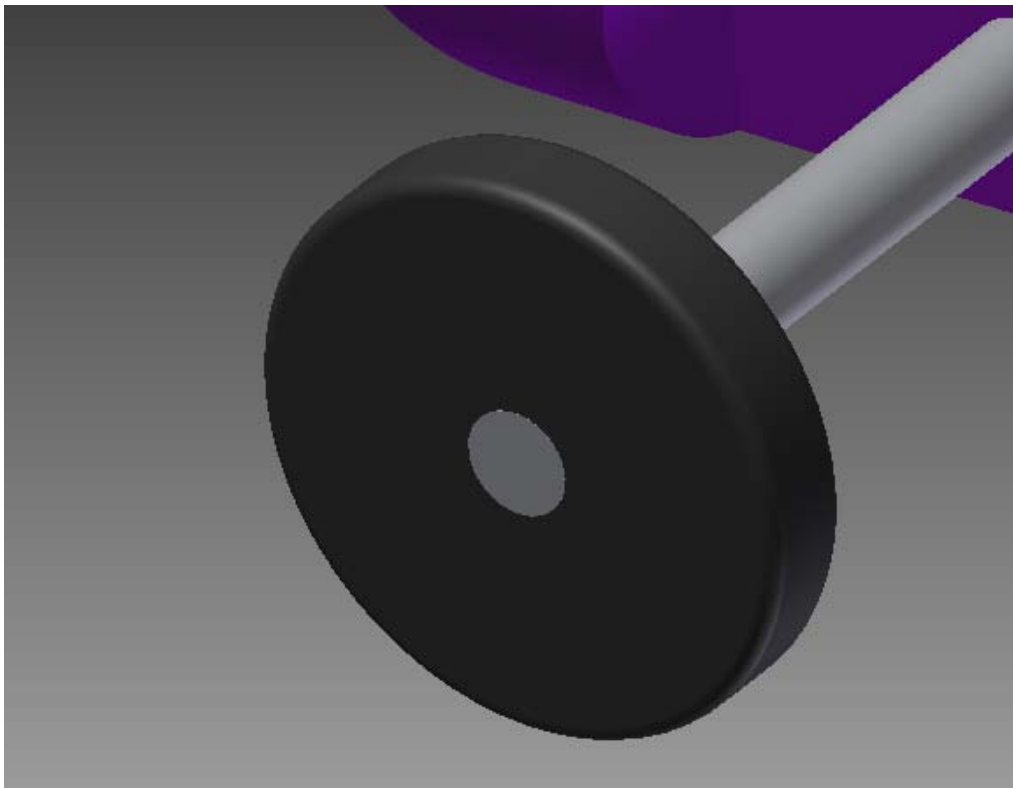
8. Place *four wheels* in the **Graphics Window**.



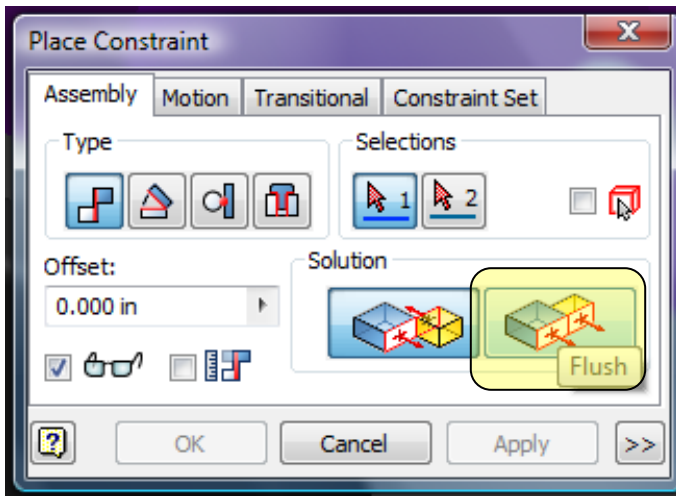
9. Use the Constrain/Mate procedure that you just used to put the wheels on the axes. You will have to rotate the vehicle to get to the wheels on the opposite side.



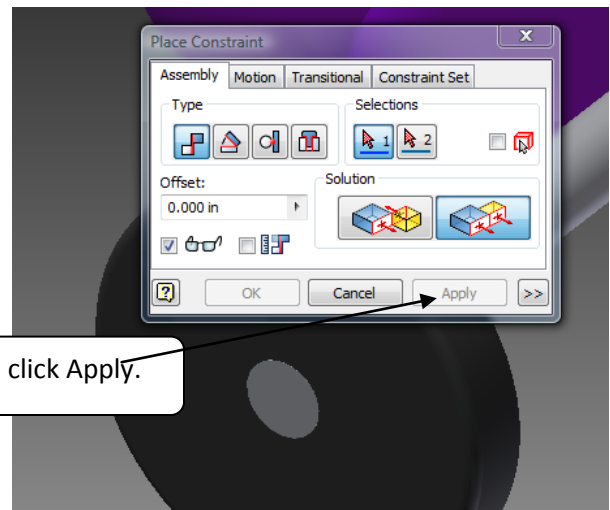
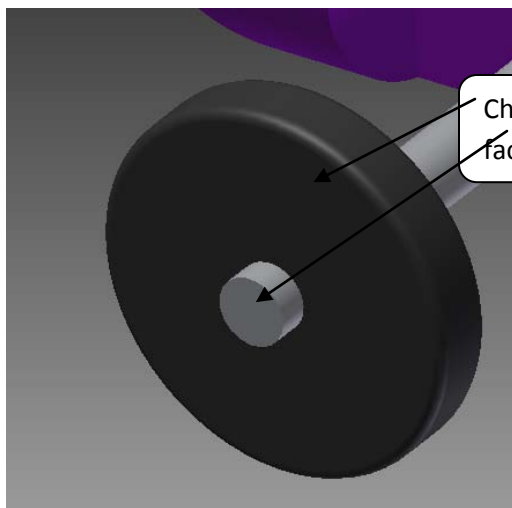
10. Next, you will create a flush mate with the wheel and axle. See the example below, then go to step 11 to complete the procedure.



11. Click on the **Assemble > Position > Constrain**. Choose **Flush** from the **Solution** area.

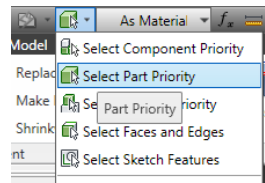
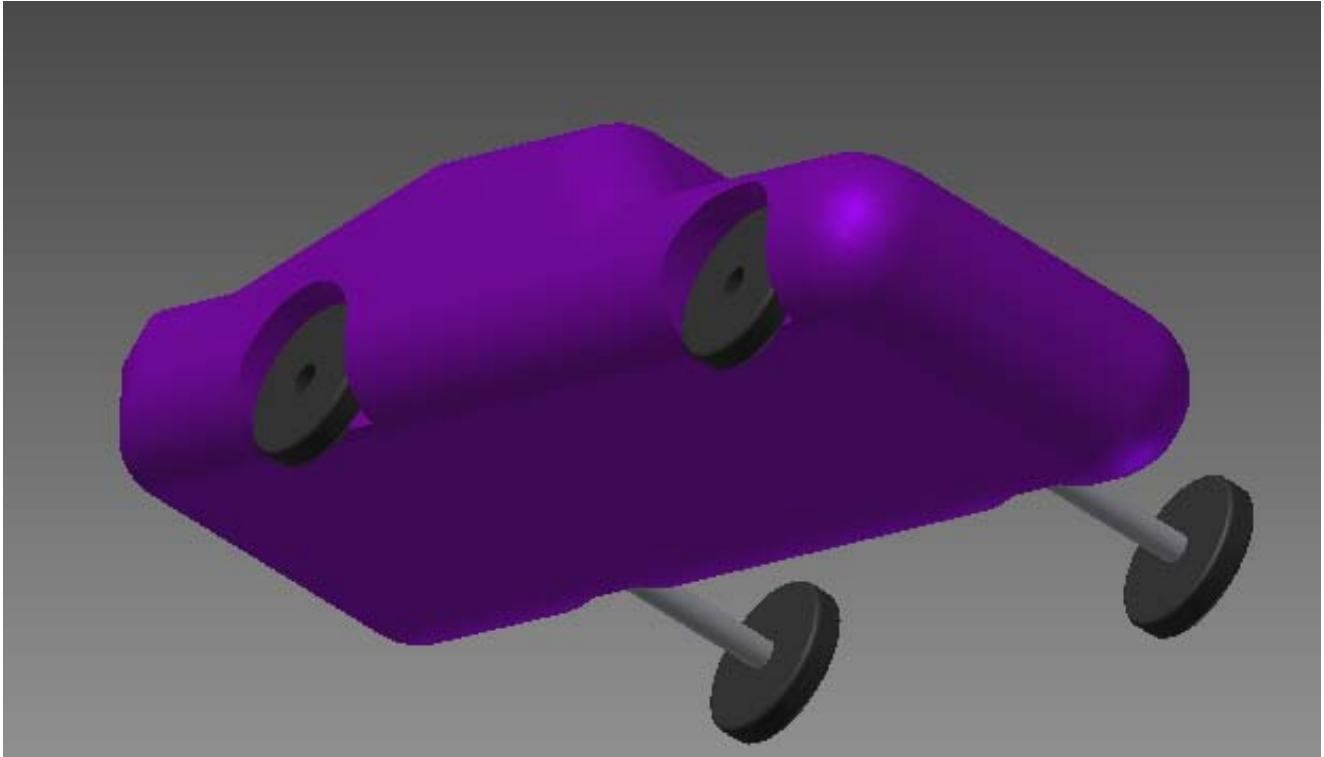


12. Choose the **flat** face of the **axle** and the **flat** face of the **wheel**. Click Apply.



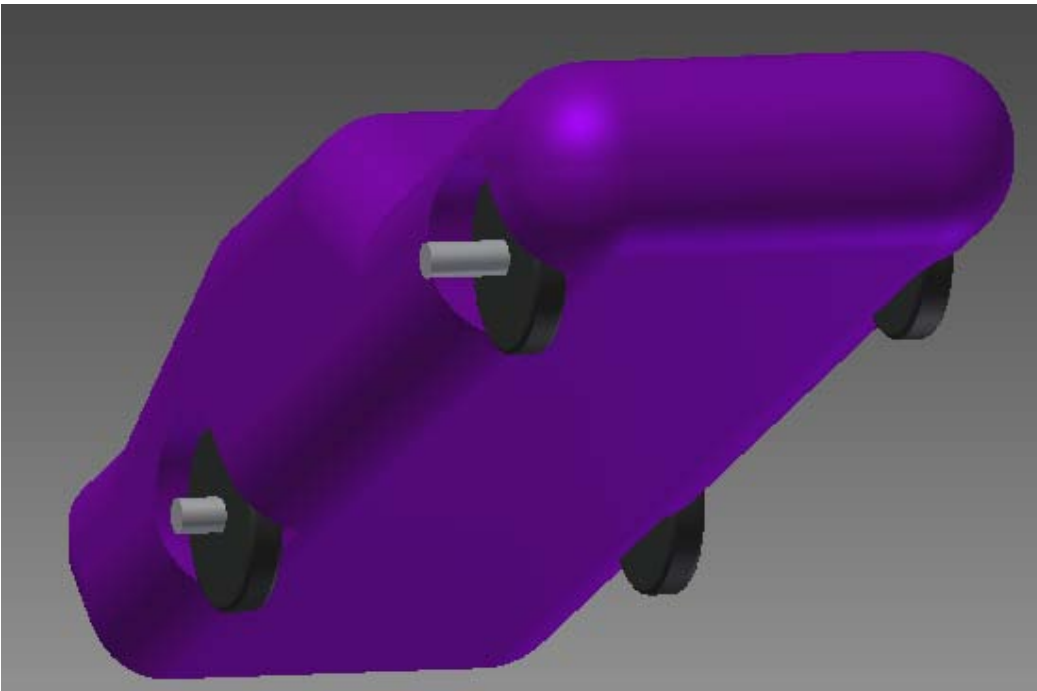
13. With the Place Constraint box still active, repeat the procedure for the other wheel that is on the same side. Click Cancel when finished with both wheels.

14. Rotate the vehicle so that you can see the other side.

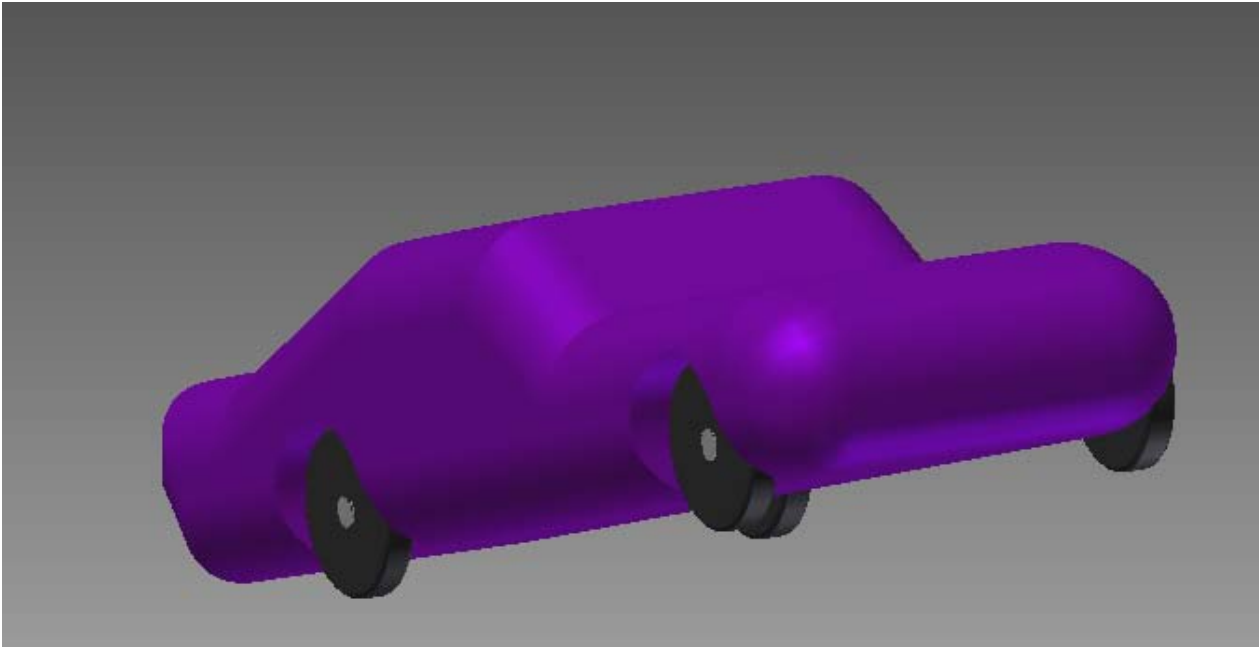


15. Select Part Priority from the Select menu. Drag the axles through so that they protrude from the body.

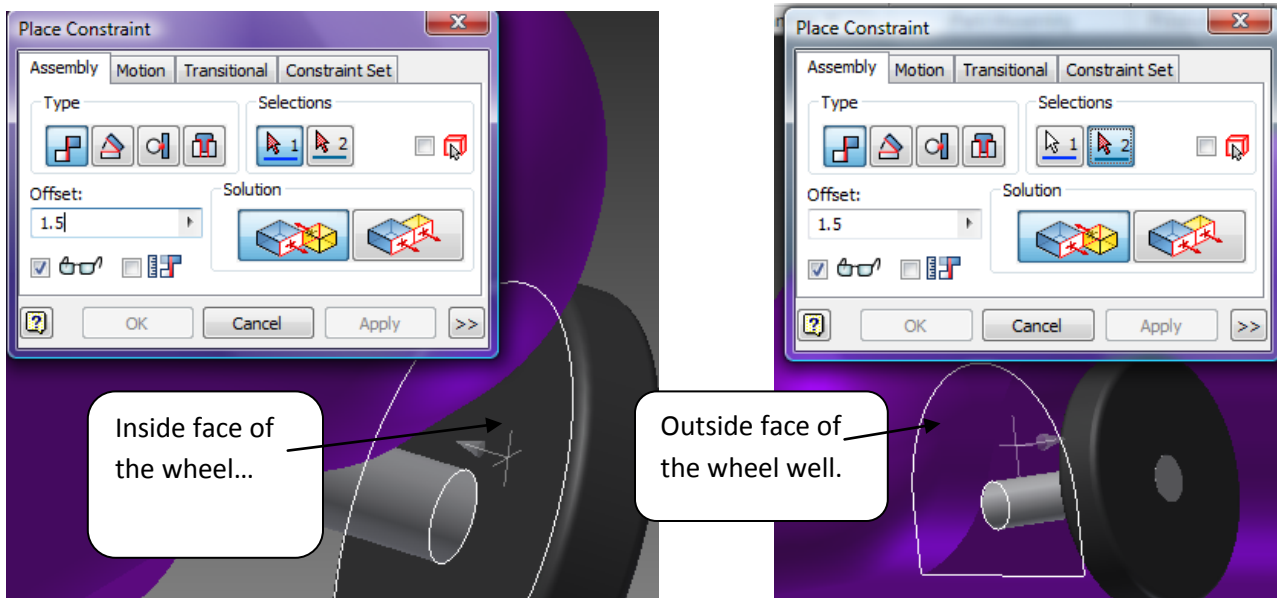
Drag the axles through so that they protrude from the body.



16. Repeat the Flush Constraint procedure from steps 11 through 15.



17. Next, you will create an *offset mate* between the wheel and the inside of the wheel well. Select the **Constrain** tool and the **Mate** solution. Set the Offset to 1.5. Choose the *inside face* of the **wheel** and the *outside face* of the **wheel well**. Click **Apply**. See Below:



18. Repeat the offset mate procedure for the other *Axle*. **DO NOT** OFFSET THE OPPOSITE SIDE OF EACH AXLE!! THAT WILL CREATE AN OVERCONSTRAINT ERROR!!

