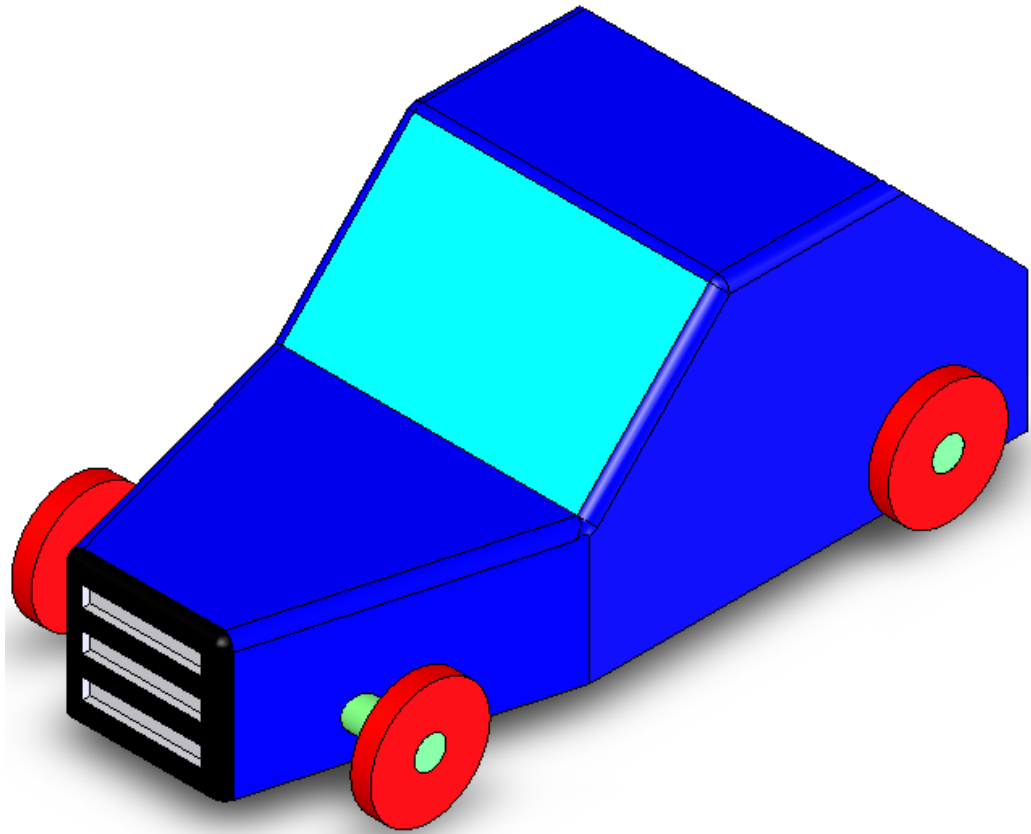


BEACH BUGGY EXERCISE 1.



Prerequisite knowledge	Students should have completed Exercises from the file: <u>Introduction to Assemblies – Concept Mates</u>
Focus of lesson	This lesson will focus on using the following commands Assemblies and Mates.
Commands Used	This lesson includes <i>Sketching</i> , <i>Extruded Boss/Base</i> , <i>Extruded Cut</i> , <i>Chamfer</i> , <i>Fillet</i> and <i>Mates</i> .

Getting Started

This model consists of three separate components. Each component will be drawn separately and assembled in the Assembly screen. When creating part files which will be used to create an assembly you must ensure that each of the part files are saved in the same folder. Create a folder in My Documents and name it **Beach Buggy**. Each of the part files **must** now be saved in this folder. Saving these files in separate folders would prevent Solidworks from opening the Assembly file at a later date.

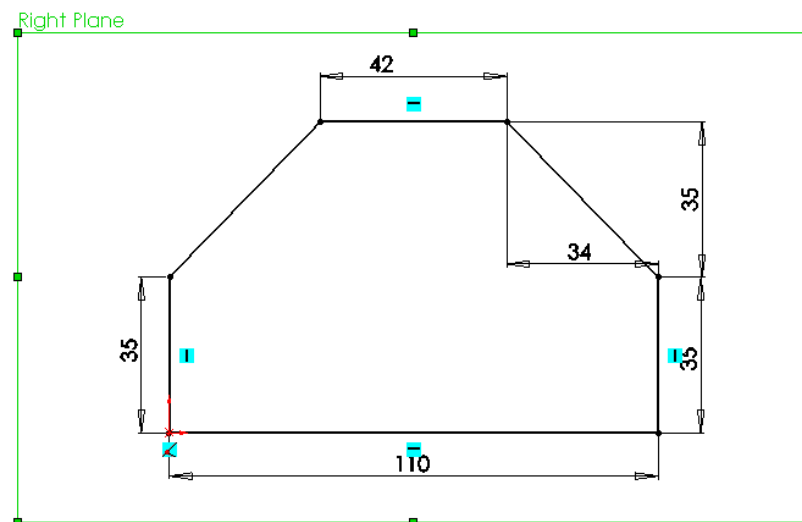
Part One: Body

Save File

Select **File, Save As**, Filename Car Body.

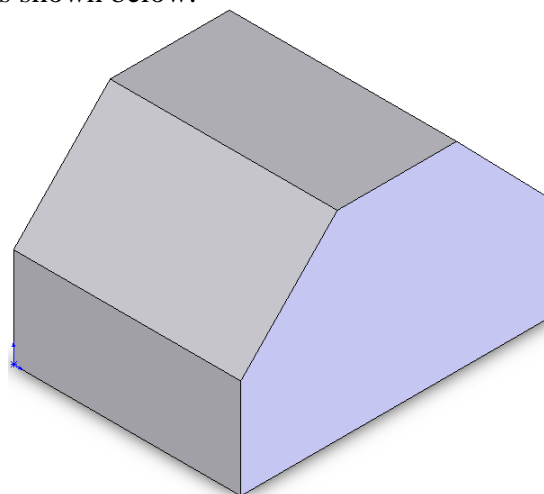
Creating the Sketch:

Select the right plane. Create and dimension a sketch as shown.



Creating the Feature:

Extrude the sketch to a depth of **80mm** as shown below.

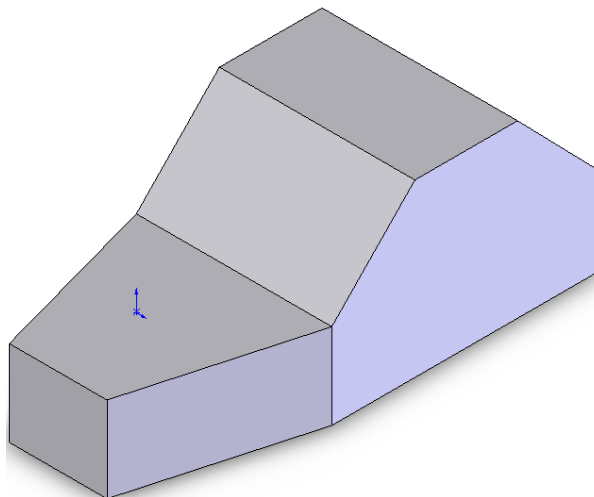
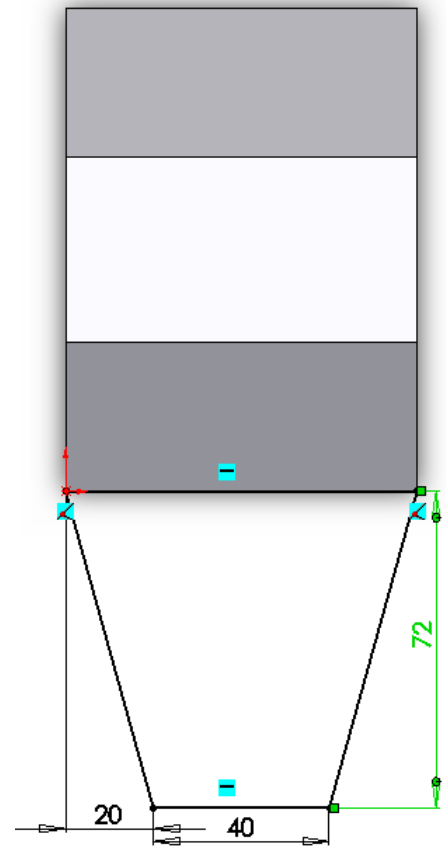


Creating sketch:

Select the Top Plane. Select Top view.
Create and dimension the sketch in the
position shown.

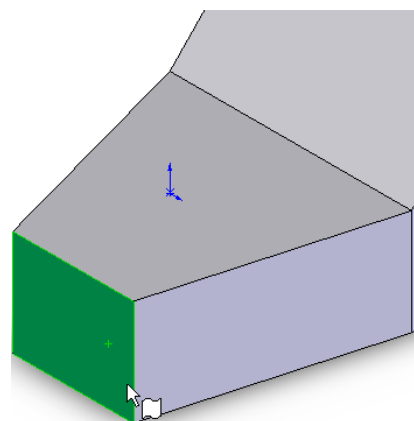
Extrude the Sketch:

Extrude the sketch to a height
of **35mm** as shown.



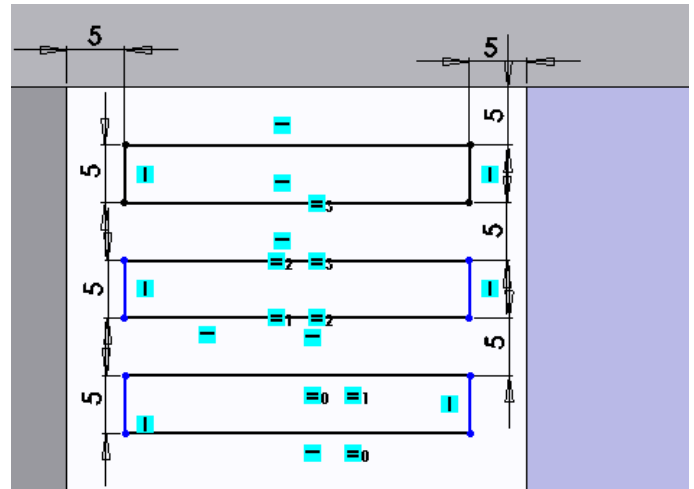
Creating Sketch:

Select the front face of the
buggy as shown



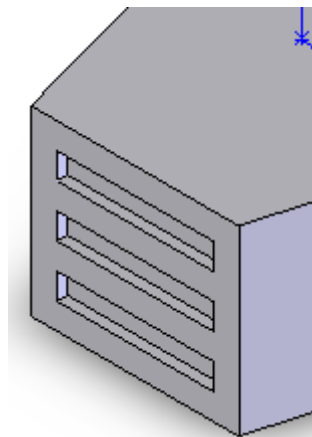
Create and dimension a sketch on the selected surface as shown.

As in Exercise Two add relations between the lengths of each of the rectangles.



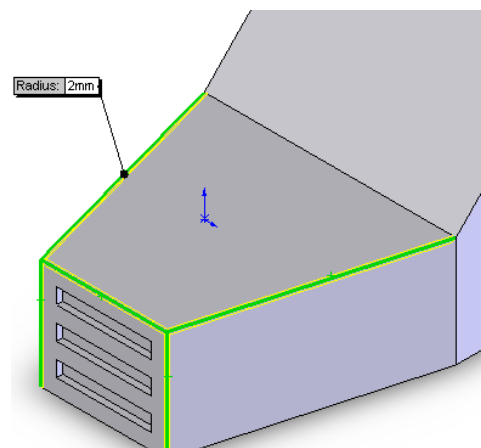
Extrude Cut:

Select Extrude Cut and select **2mm** as the depth of cut.



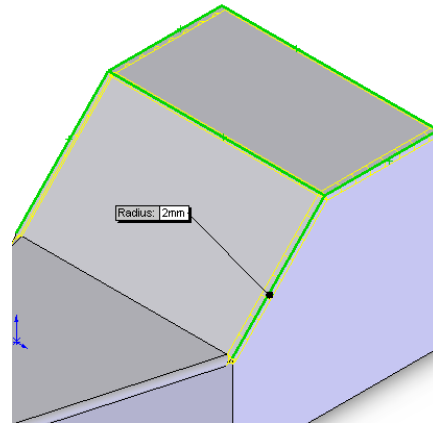
Fillet:

Create a **2mm** fillet on each of the edges as shown.



Fillet:

Now create a second set of **2mm** fillets on the surfaces shown.

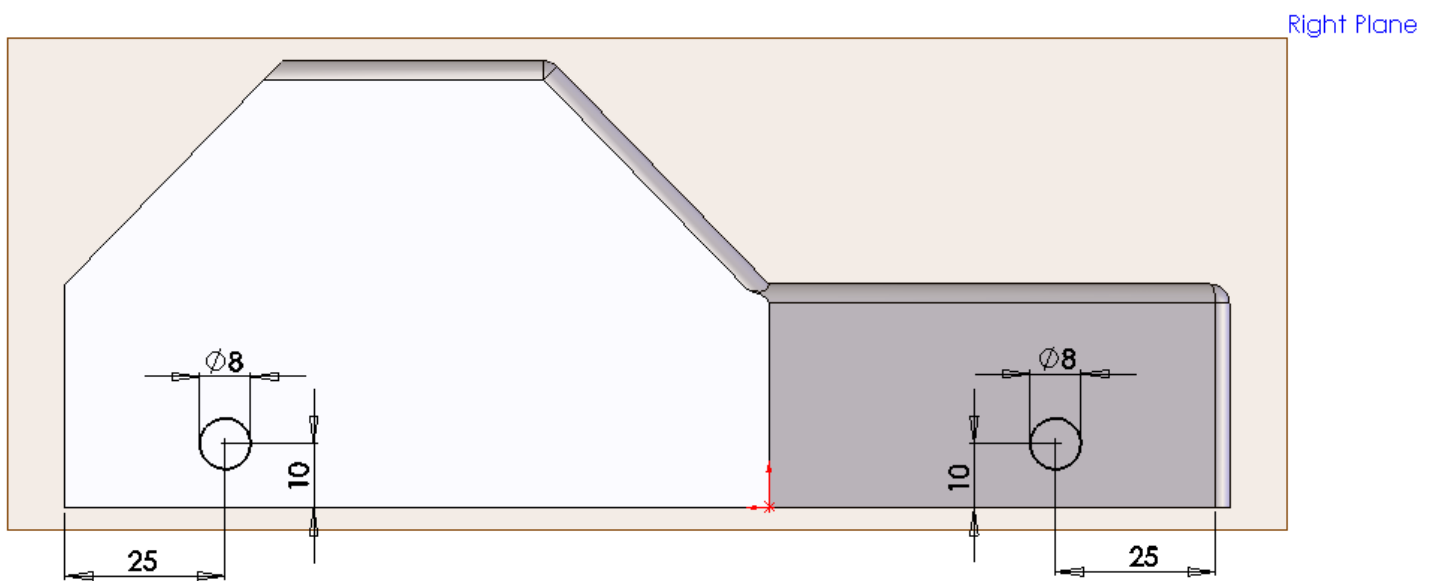


Note:

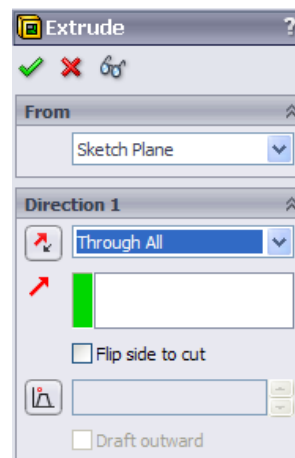
We must create these fillets in two steps because of the inclined edges.

Creating the Sketch:

For the next sketch select the Right Plane, select a left view and create and dimension a sketch of two circles as shown.

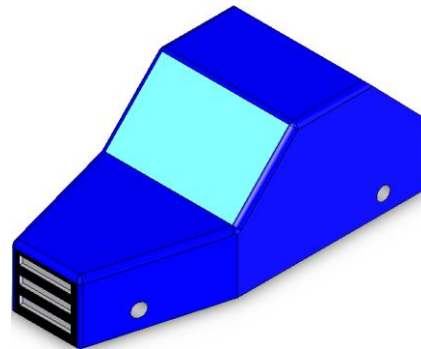


Creating Feature: Select Extrude Cut and create a cut **Through all** as shown.



Applying Colour:

Apply colour as in previous exercises as shown and then **save** file again.



Close File:

Close the file **Body** and open a new Part file.

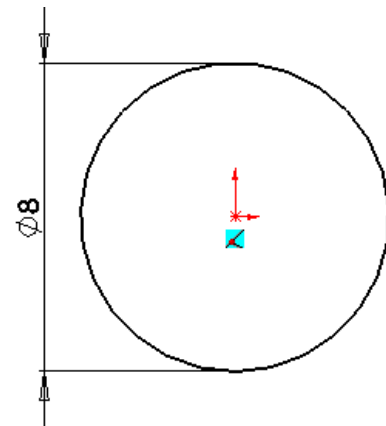
Part Two: Axle

Save File

Select **File, Save As**, Filename Axle.

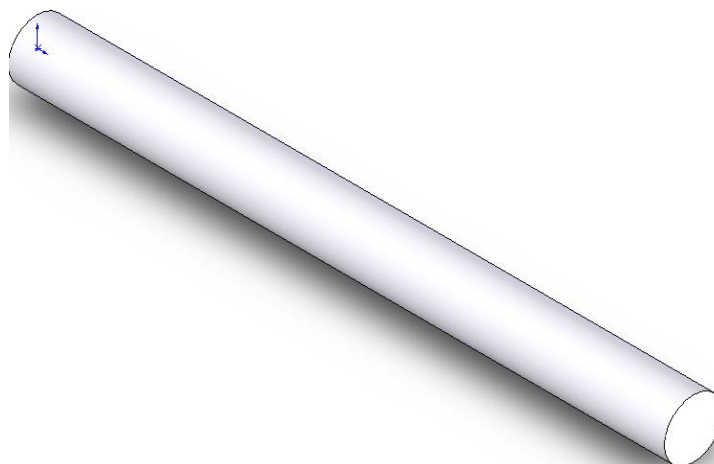
Creating the Sketch:

Select the Right Plane. Create and dimension a sketch of a circle as shown.



Creating Feature:

Extrude Boss/Base the sketch to a depth of **90mm** as shown.



Applying Colour: Apply colour as in previous exercises as shown and then **save** file again.

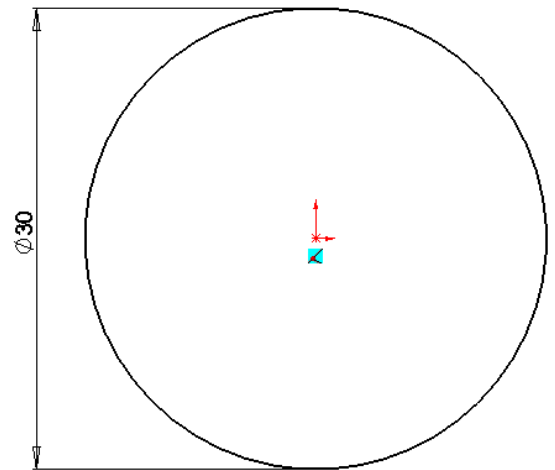


Close File: Close the file **Axle** and open a new Part file.

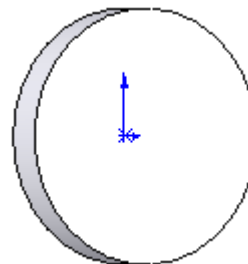
Part Three: Wheel

Save File Select **File, Save As**, Filename Wheel.

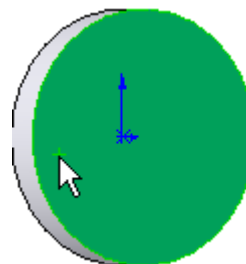
Creating the Sketch: Select the Right Plane. Create and dimension a sketch of a circle as shown.



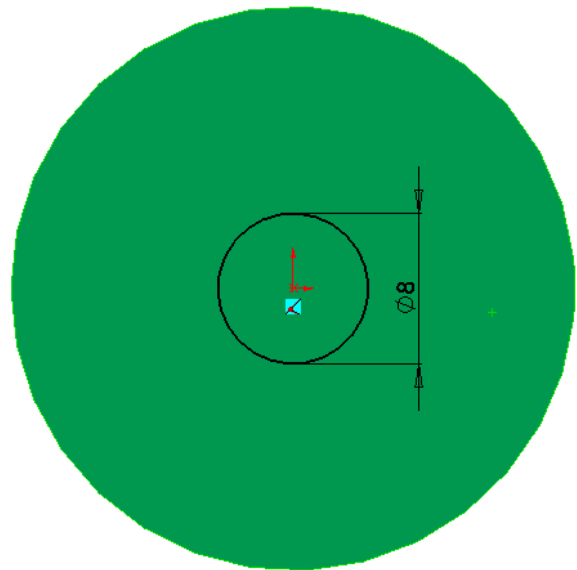
Creating Feature: Extrude Boss/Base the sketch to a depth of **5 mm** as shown.



Creating the Sketch: Select the front face of the wheel as shown.

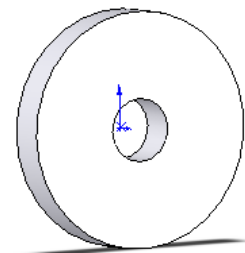
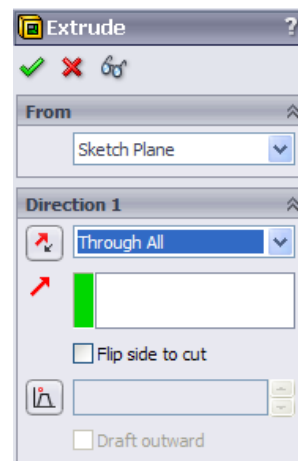


Create and dimension a sketch of a circle from the centre of the selected surface as shown.



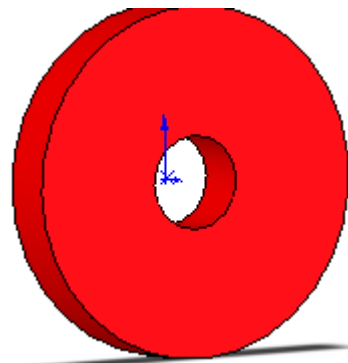
Creating Feature:

Extrude Cut the sketch **Through all** as before.



Applying Colour:

Apply colour as in previous exercises as shown and then **save** file again.



Close File:

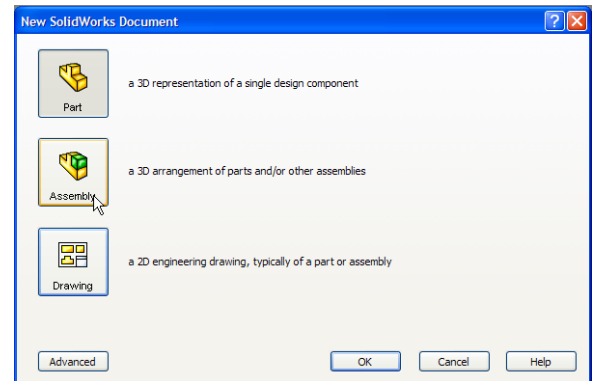
Close the file **Wheel**.

Assembly: Assemblies are used to assemble or join two or more Solidworks Part files.

Opening Assembly Files:

Select **File** and **New** from the toolbar at the top left hand corner of the Solidworks screen.

From the dialogue box which appears select **Assembly** and click OK as shown.



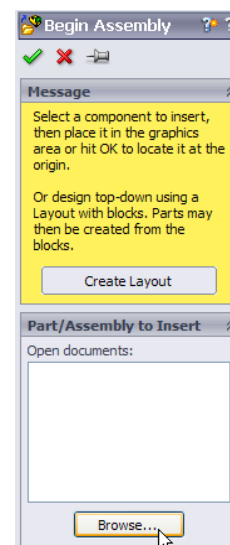
Solidworks will now bring you into the Assembly screen. This screen is similar to the part file screen with some important differences. These differences will be highlighted as we create the assembly.

Save File: Select **File**, **Save As**, Filename Beach Buggy.


Note: Assembly files will save as **.ass** files.

Getting Started: On the left hand pane of the screen you will see a dialogue box titled 'Insert Component'. This box will allow you to bring in the beach buggy part files that you have just created.

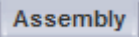
From this Dialogue box select the Browse button as shown.

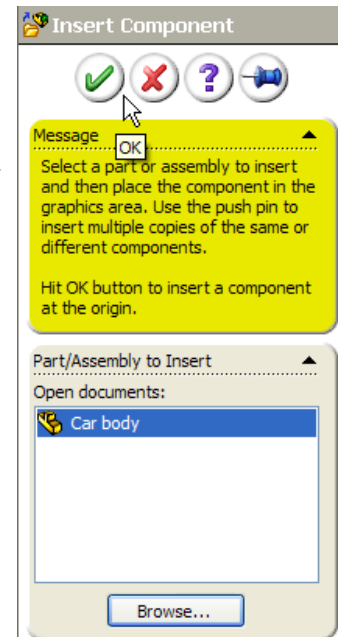


Browsing for Part Files: When you select browse Solidworks will automatically direct you to the last folder you saved work into, in this case **Beach Buggy**. From this folder select the **Car body** file as shown and select Open

You will now be brought back into the Solidworks screen. In the Insert Component box the file Car Body should now appear. Click the  OK button to position the model.

The Car Body is now visible on your screen.

Note: One of the differences you will notice when creating Assemblies is that the Features button has been replaced with an Assemblies button  in the top left hand corner.

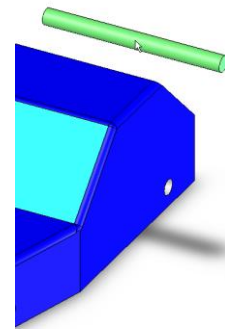


The assemblies' button offers all the options needed to create an assembly.

Insert Axles: To insert the next part file, select Assemblies Button and then select Insert Components.



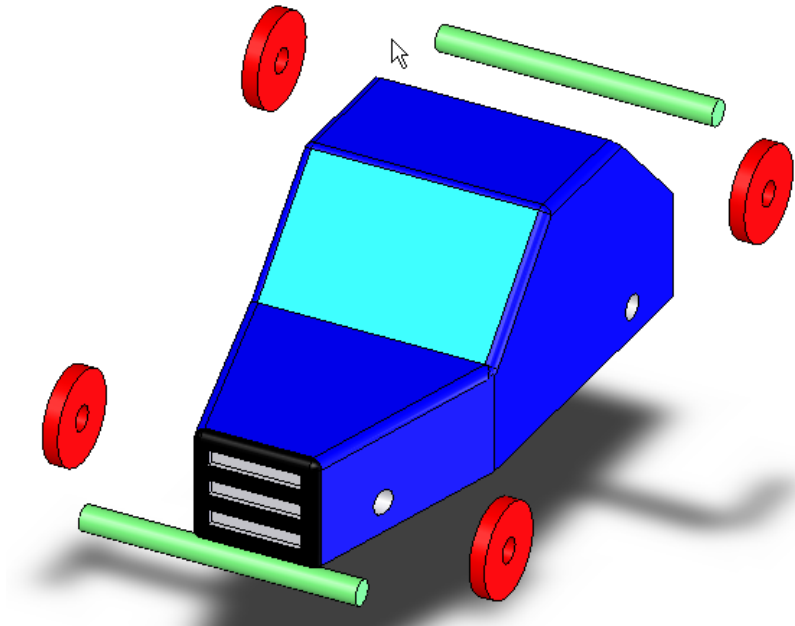
On the left hand pane of the Solidworks screen the insert Components dialogue box will appear. Click Browse and insert axle from the **Beach Buggy** folder. To place the axle on the screen left click near the back of the car body, as shown.




Note: To move a component in an assembly left click on it, holding down the left mouse button drag it to a more suitable area of the screen.

Note: As there are two axles the last step will have to be repeated to bring in the second axle.

Insert Wheels: To insert wheels repeat the method used to insert the axles. Again as there are four wheels the step will have to be carried out four times. Your assembly screen should now resemble the one shown below.

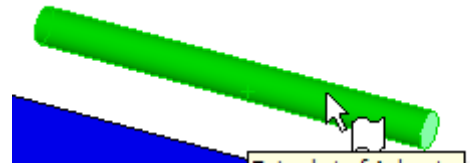


Introducing Mates: Mates are used in assemblies to join components together. Every time two components are joined together, Mates, will be used. Mates add relationships between components and aligns surfaces or edges. To create a mate click the assemblies button and the select 

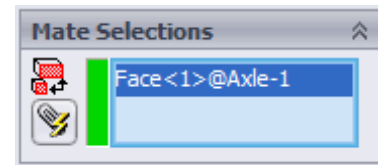
Creating Mates: The first mate we will create will be to position the axle at the back of the Car Body.

Mate:- Click the mate button from the assemblies toolbar. On the left hand pane of the Solidworks screen the mates dialogue box will appear. The pink mate selections box which is at the top of the mates dialogue box is used to display the selected parts, edges or surfaces.

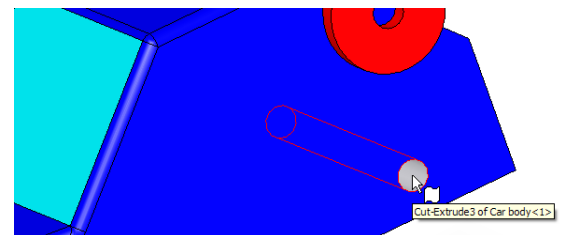
To begin with select the outside surface of the axle as shown.



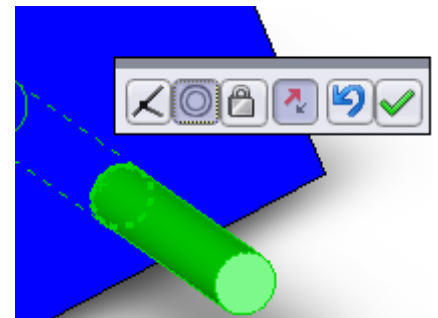
The blue mate selections box will now register this face as shown.




Next select the axle hole at the back of the car body as shown.



Solidworks will now automatically make the axle and the axle hole concentric as shown.



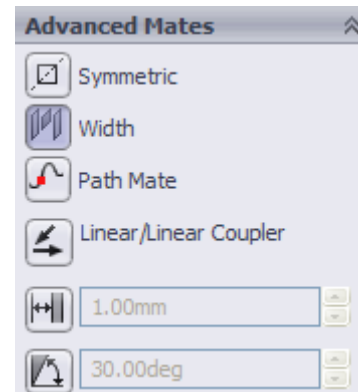
A new dialogue box appears on the screen as can be seen above. This box allows you to change the mate used or to confirm the selected mate with the OK  button.

Now that the axle is in position the next step is to centre it with the car body. To do this we will have to use a mate called **Width Mate**.

Introducing Width Mate: Width mate is part of an optional set of mates called **Advanced Mates**. Width mate as the name suggest is used to set a specific width between two parts. In this case we will use the **Width Mates** to set the distance between the end of the axle and the car body.

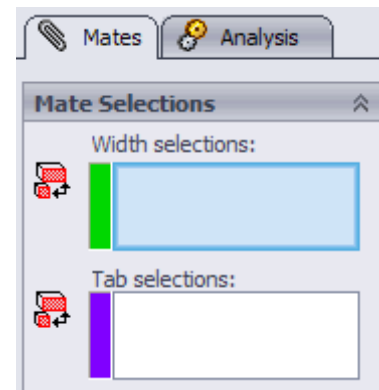
Where to find it:

Click on the Mate icon. On the mates dialogue box select **Advanced Mates** and **Width Mate** as shown.

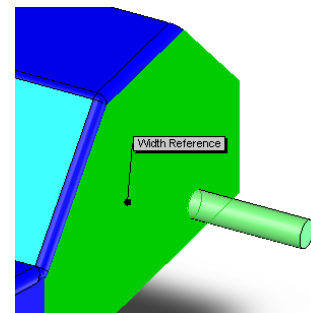


Using Width Mate:

When you select the Width Mate option from Advanced Mates, Solidworks will ask you to select the surfaces you wish to set distance between and also to set the distance required.

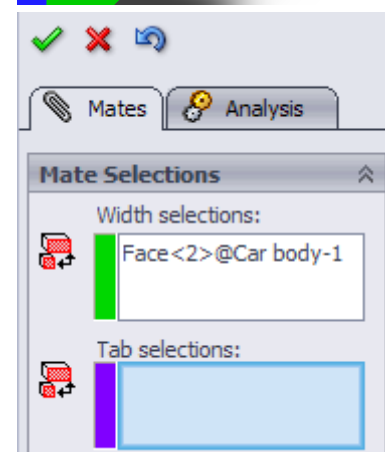


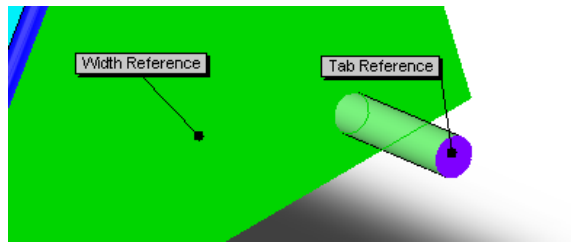
The box marked Width selection will contain the surface from which you wish to set the distance from, in this case the **Car Body**. Left click the car body as shown to select it.



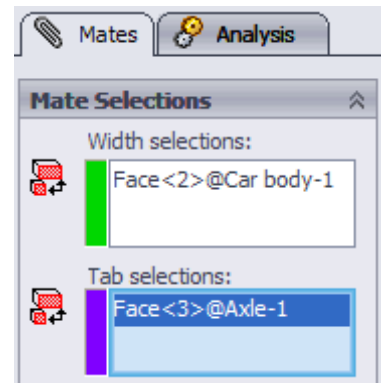
Next we must select the tab, this is the surface or edge which will be placed a set distance away from the car body in this case the axle. First left click in the **Tab selections** box as shown. Note how the box now changes to a pink colour.

Now left click on the end face of the axle as shown.




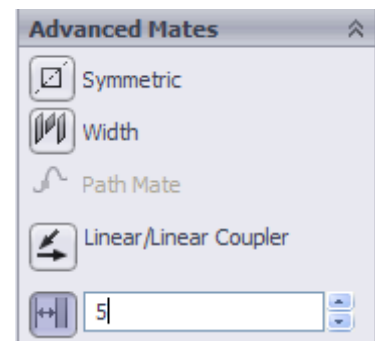


The **Tab selections** box will now contain the axle face you have selected as shown.

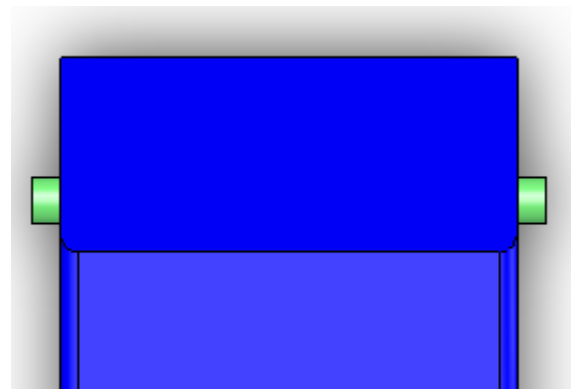


Both faces have now been selected so the next step is to select the distance.

In advanced mates click on the distance button to activate the distance and change the value to **5mm** and click OK 

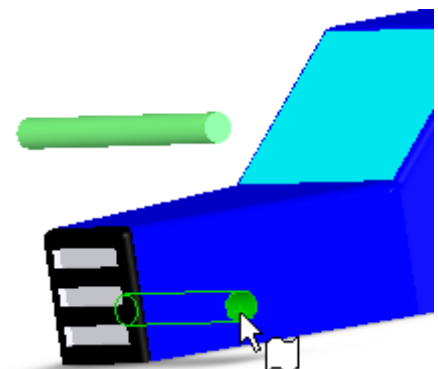


The end of the axle will now move to within **5mm** of the car body as shown.

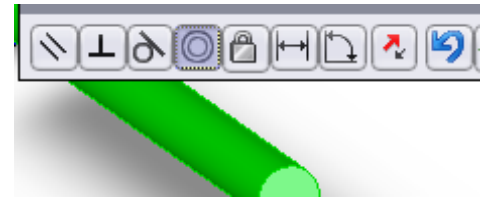


To mate the front axle to the front of the car body we will repeat the steps used for the front axle.

Select Mates, now select the front axle and the front axle slot in the car body.



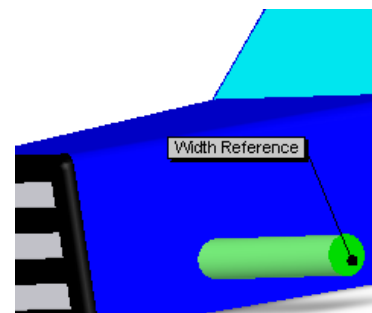
SolidWorks will automatically add a concentric mate to the axle and slot as shown.



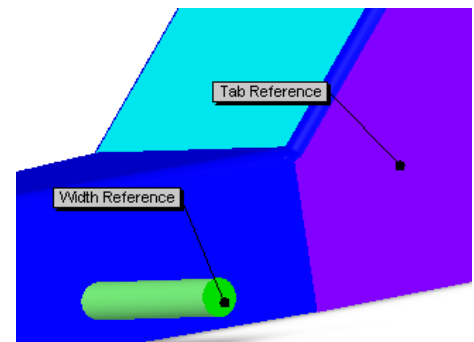
We will now add a width mate to the axle and car body. This time, however, we will not be able to use the side surface at the front of the car as it is set at an angle.

To create the width mate, select **Mates** and as before select **Width Mates** from the advanced mates option.

Select the end face of the axle as the width reference, as shown.

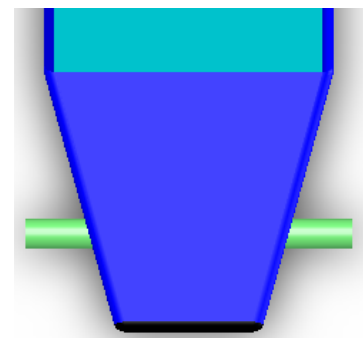


Next, select the straight side surface of the car body as shown.



Finally, select the distance button in advanced mates, as before, and set the distance to **5mm**.

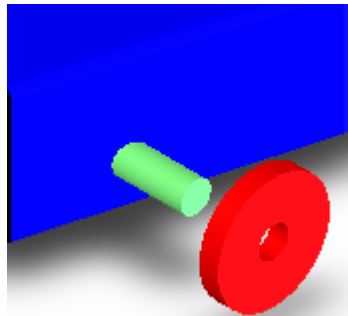
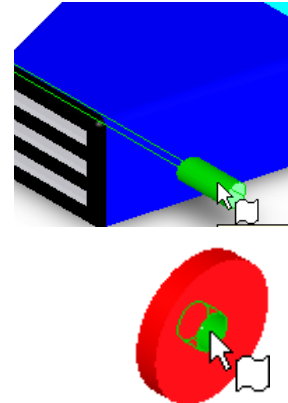
Selecting a “Top View” will allow you to see the position of the axle.



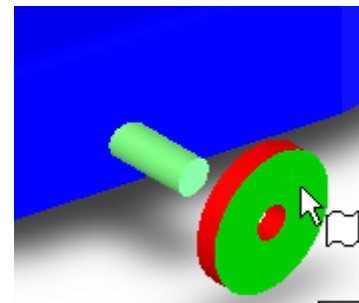
The next step is to mate the wheels to the axles.

Select the **Mates** icon from the assemblies toolbar.

Next select the outside surface of the axle and the inside surface of the hole through the wheel and click OK. This will make the wheel and the axle concentric, as shown.

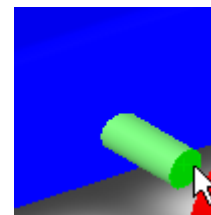


To finish this mate select the outside surface of the wheel, as shown

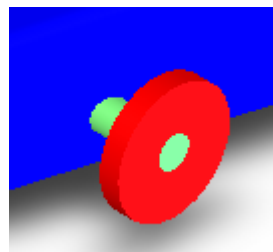


Now select the end surface of the axle, as shown.

Click OK  to finish mate.



This will align the two selected surfaces.



Repeat these steps for each of the remaining three wheels.

Finished Assembly:

