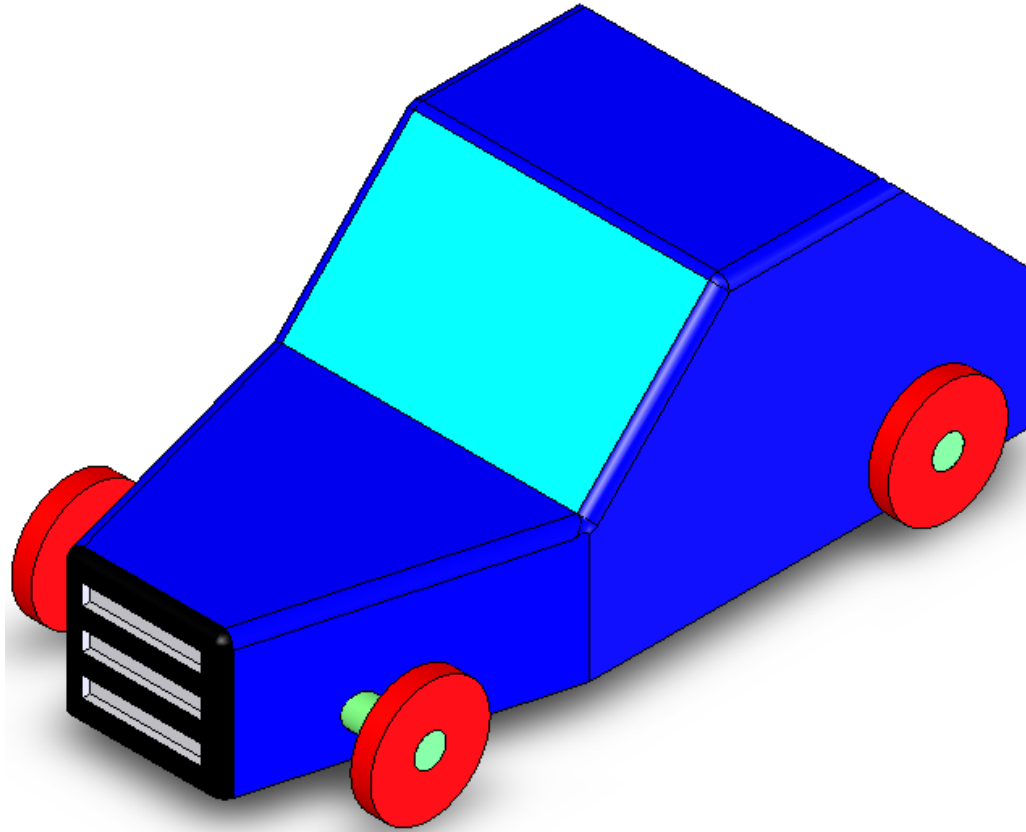


## EXERCISE ONE: BEACH BUGGY.



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

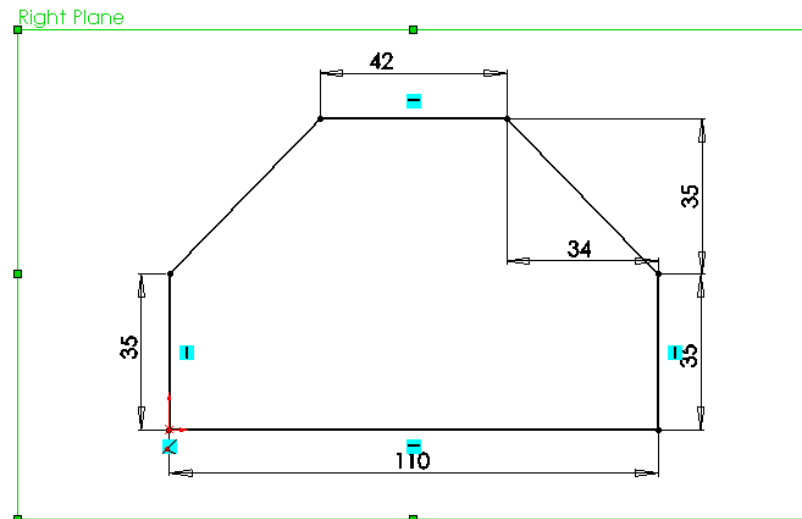
### Part One: Body

#### Save File

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

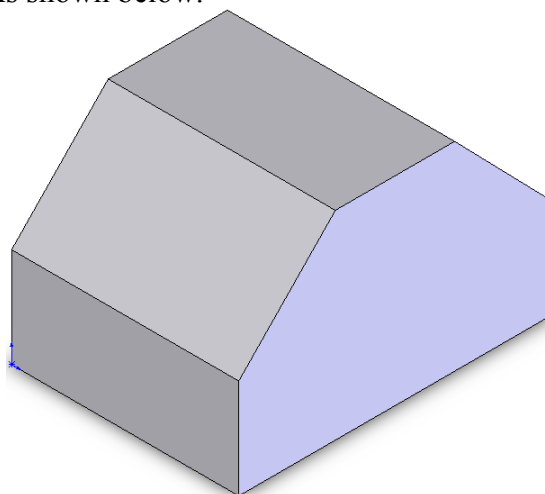
#### Creating the Sketch:

Select the Right plane. Create and Dimension a sketch as shown.



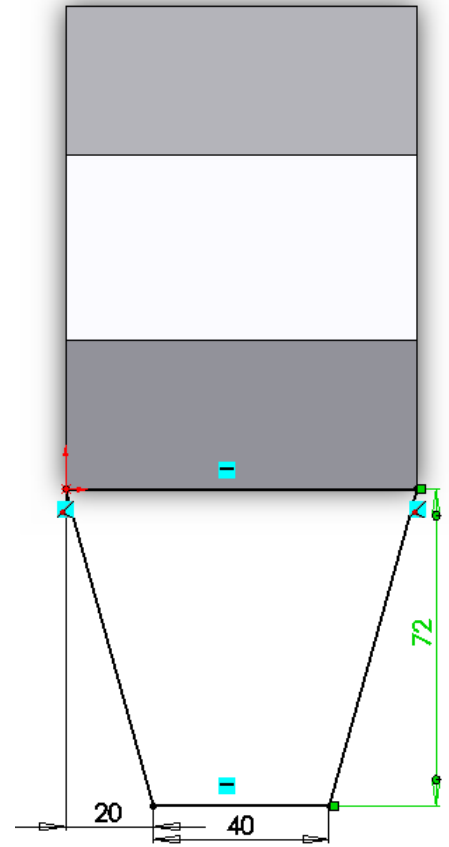
#### Creating the Feature:

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



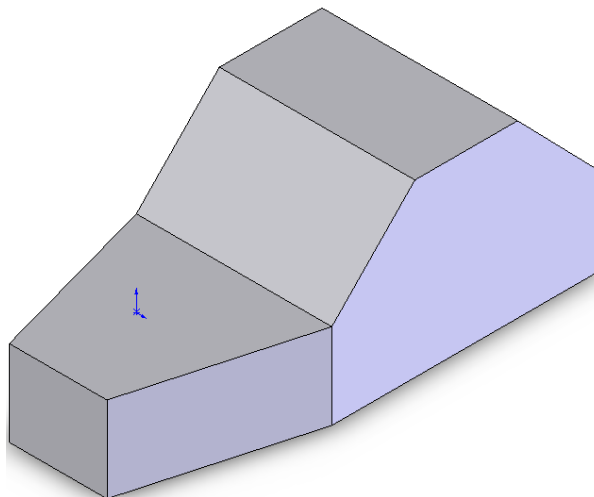
**Creating sketch:**

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



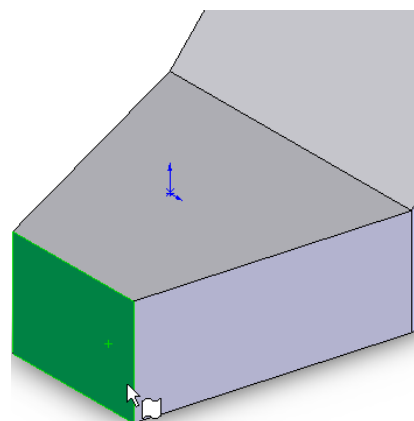
**Extrude the Sketch:**

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



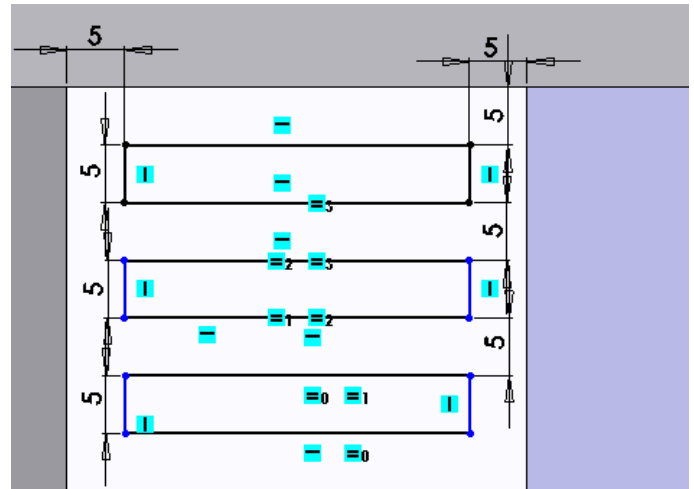
**Creating Sketch:**

Select the front face of the  
buggy as shown



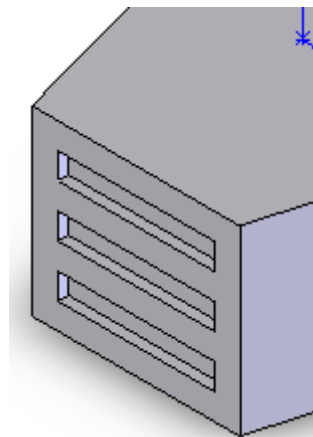
Create and dimension  
A sketch on the selected  
Surface as shown.

As in Exercise Two  
Add relations between  
The lengths of each of the  
Rectangles.



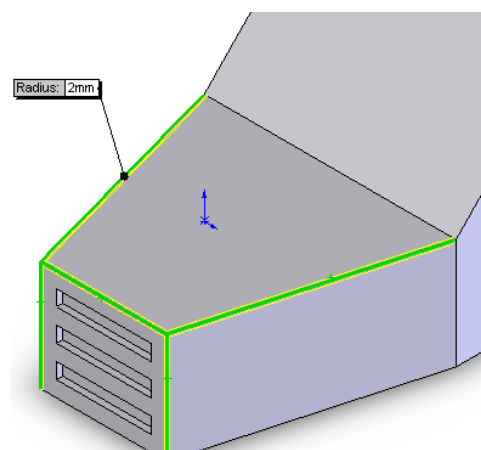
**Extrude Cut:**

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



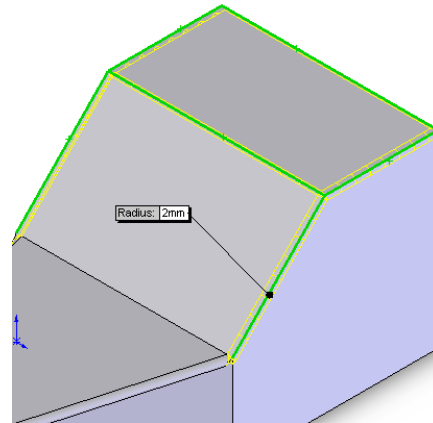
**Fillet:**

Create a **2mm** fillet  
On each of the edges  
As shown.



**Fillet:**

Now create a second  
Set of **2mm** fillets  
On the surfaces shown.

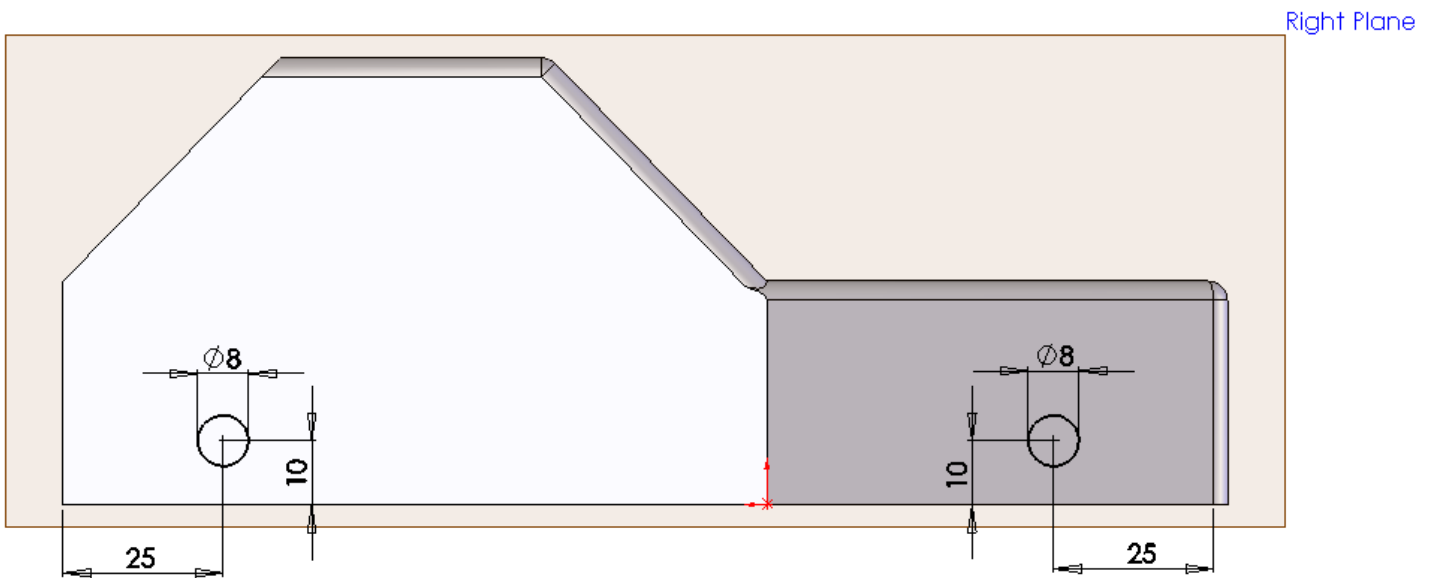


**Note:**

We must create these fillets  
In two steps because of the  
Inclined edges.

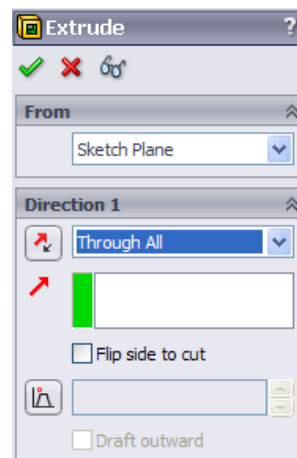
**Creating the Sketch:**

For the next sketch select  
The Right Plane, select a  
Left view and create and  
Dimension a sketch of two  
Circles as shown.



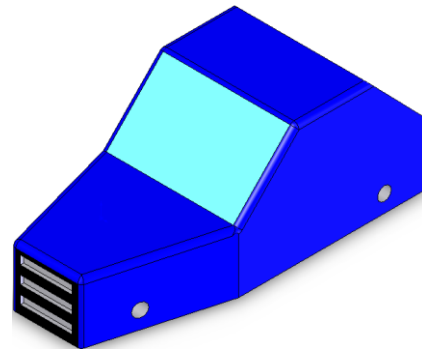
**Creating Feature:**

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.

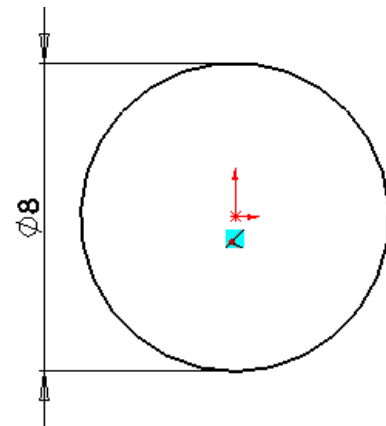
**Part Two: Axle**

**Save File**

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

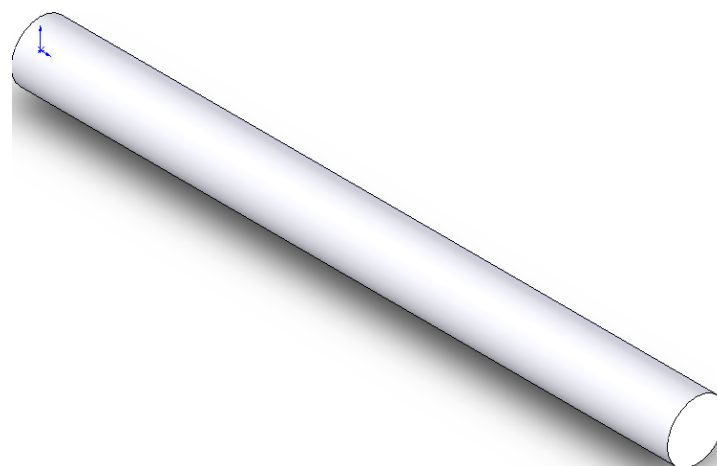
**Creating the Sketch:**

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.

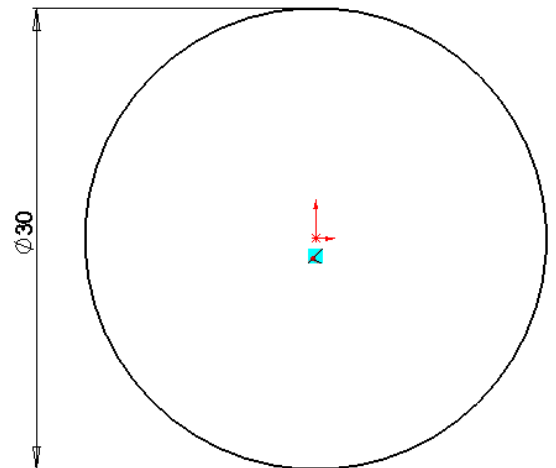


**Close File:** Close the file **Axle** and open a new Part file.

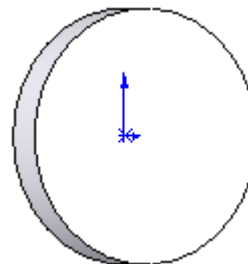
### Part Three: Wheel

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

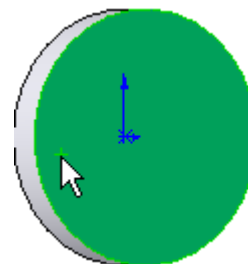
**Creating the Sketch:** Select the Right Plane. Create and Dimension a sketch of a circle as Shown.



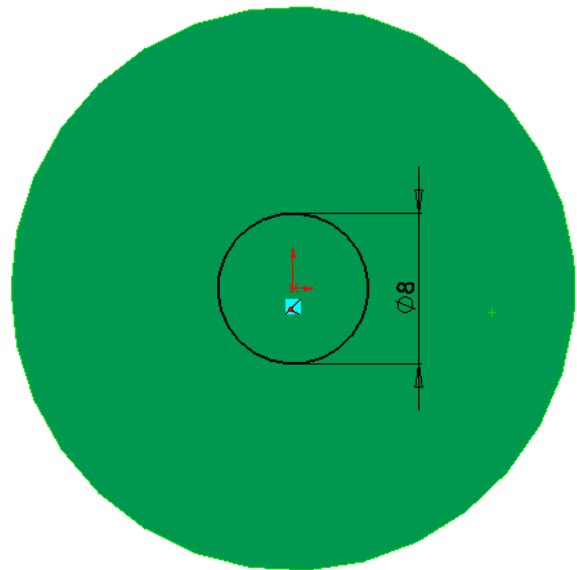
**Creating Feature:** Extrude Boss/Base the Sketch to a depth of **5 mm** as shown



**Creating the Sketch:** Select the front Face of The wheel as shown

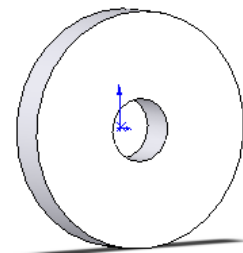
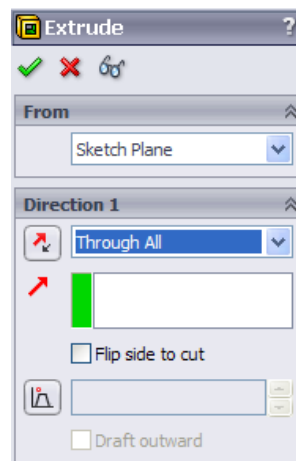


Create and dimension a  
Sketch of a circle from  
The centre of the selected  
Surface as shown.



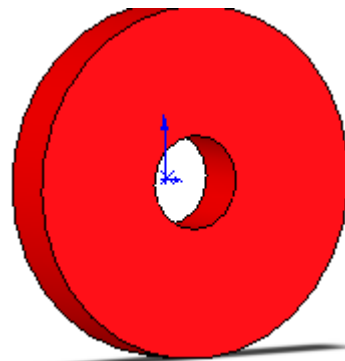
**Creating Feature:**

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**.

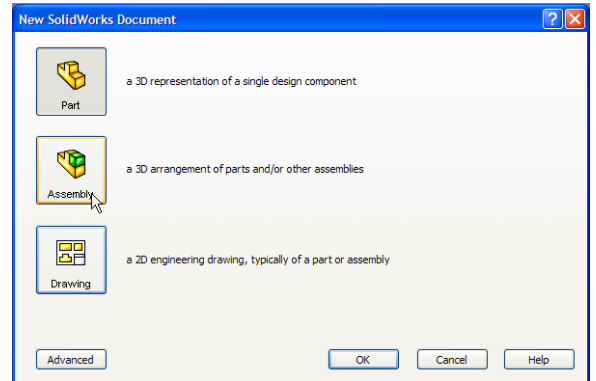


**Assembly:** Assemblies are used to assemble or join two or more Solidworks Part files.

**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.



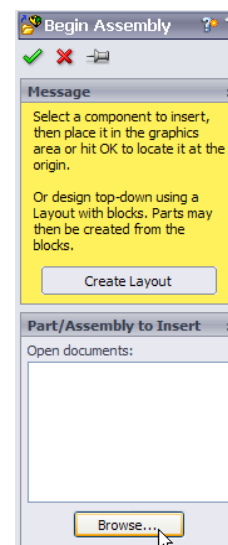
Solidworks will now bring you Into the Assembly screen. This Screen is similar to the part file Screen with some important Differences. These differences will Be highlighted as we create the assembly.

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


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

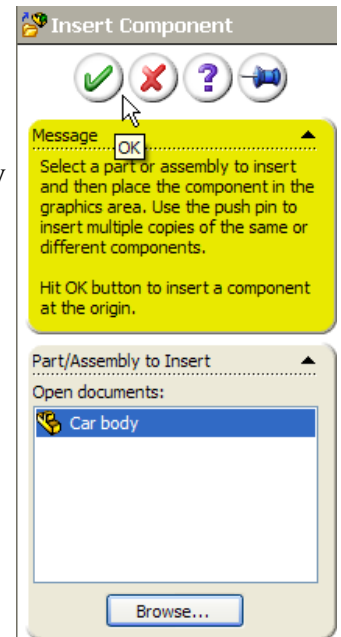
**Getting Started:** On the left hand pane of the Screen you will see a dialogue box Titled “Insert Component”. This box Will allow you to bring in the beach Buggy part files that you have just Created.

From this Dialogue box select the Browse button as shown.

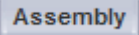


**Browsing for Part Files:** When you select browse Solidworks will automatically direct you to the last folder you saved work into, in this case **Beach Buggy**. From this folder select the **Car body** file as shown and select Open

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.

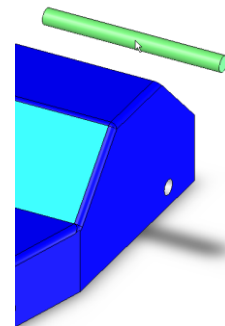
**Note:** One of the differences you will notice When creating Assemblies is that the Features Button has been replaced with an Assemblies Button  In the top left hand corner.

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 component.



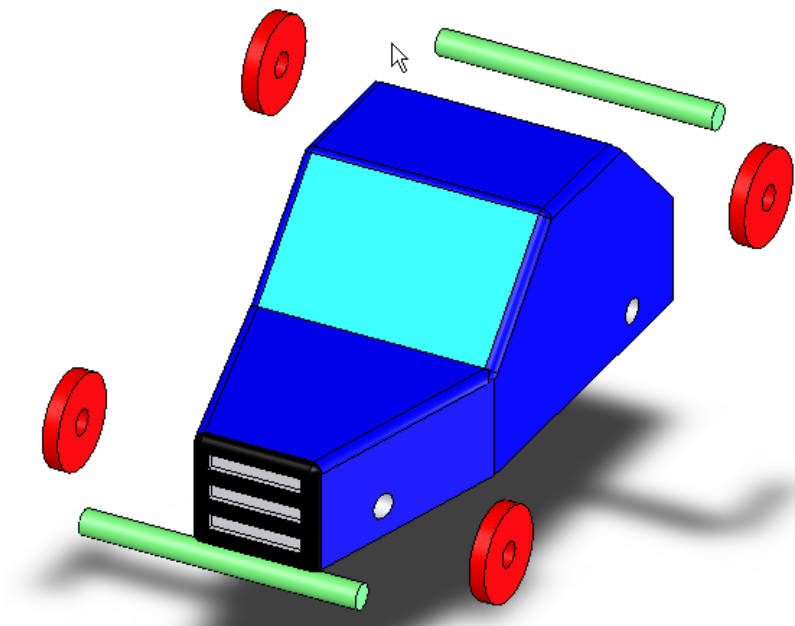
On the left hand pane of The Solidworks screen the Insert Components dialog box Will appear. Click Browse and insert Axle from the **Beach Buggy** folder. To place the axle on the screen left click Near the back of the car body, as shown.




**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.

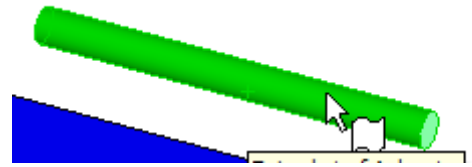


**Introducing Mates:** Mates are used in assemblies to join components together.  
Every time two components are joined together Mates will be  
Used. Mates add relationships between components and aligns  
Surfaces or edges.  
To Create a mate click the assemblies button and the select 

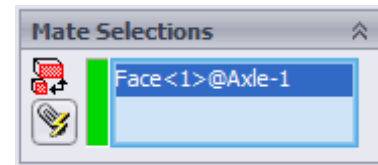
**Creating Mates:** The first mate we will create will be to position the Axle in the  
At the back of the Car Body.

**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.

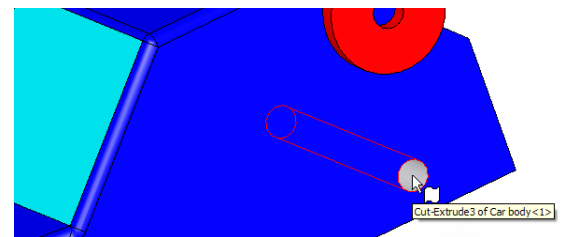
To begin with select the outside surface of the Axle as shown



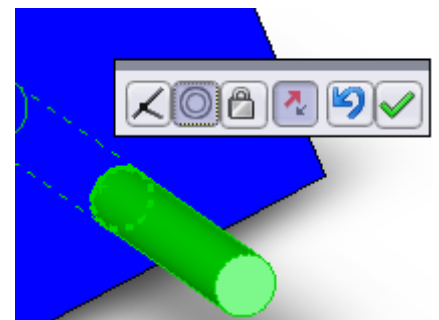
The Blue mate selections box will now register this face as shown




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.



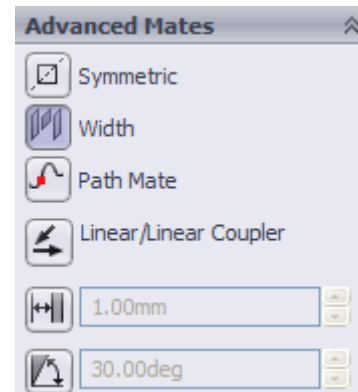
A new dialogue box appears on the screen as can be seen Above. This box allows you to change the mate used or to Confirm the selected mate with the OK  button.

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 Mate** to set the distance between The end of the axle and the car body.

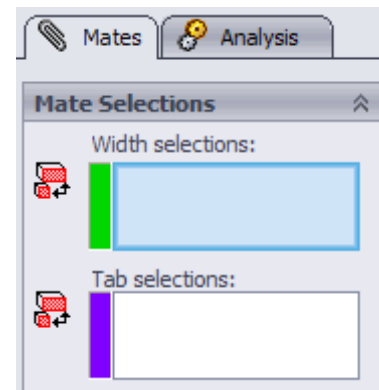
**Where to find it:**

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

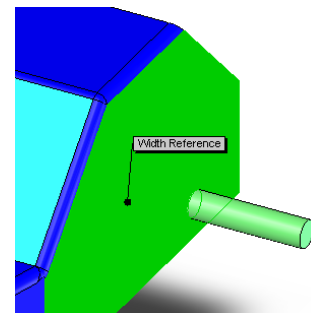


**Using Width Mate:**

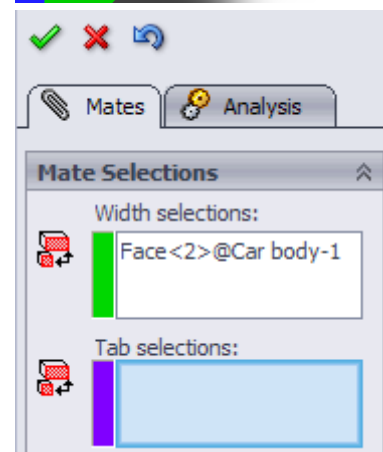
When you select the Width Mate option from advanced Mates, Solidworks will ask you To select the surfaces you Wish to set distance between And also to set the distance Required.



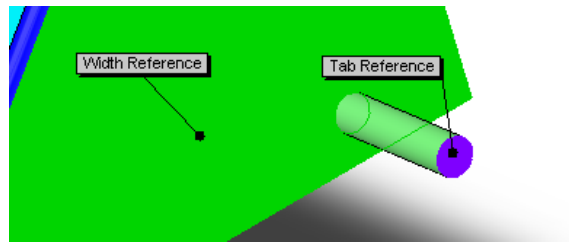
The box marked Width selection Will contain the surface from which You wish to set the distance from, in This case the **Car Body**. Left click The car body as shown to select it



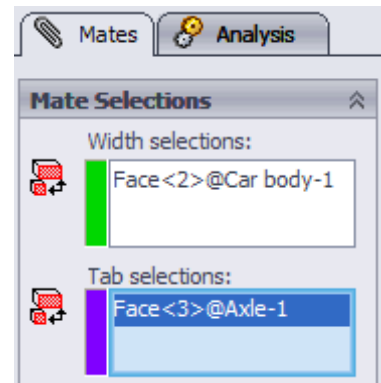
Next we must select the Tab, this is The surface or edge which will be placed A set distance away from the car body in This case the Axle. First left click in the **Tab Selections** box as shown Note how the box now changes to a Pink colour.




Now left click on the end face Of the Axle as shown.

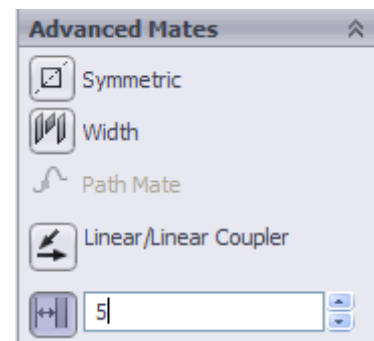


The Tab Selection box will now  
Contain the axle face you  
Have selected as shown.

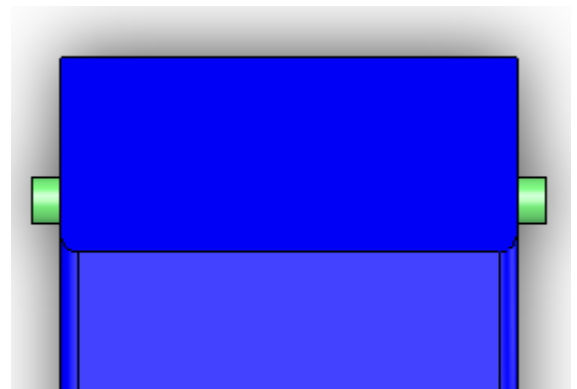


Both faces have now been selected  
So the next step is to select the distance.

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

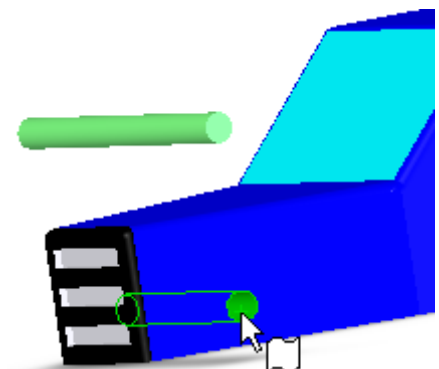


The end of the axle will now move to  
Within **5mm** of the car body as shown.



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.



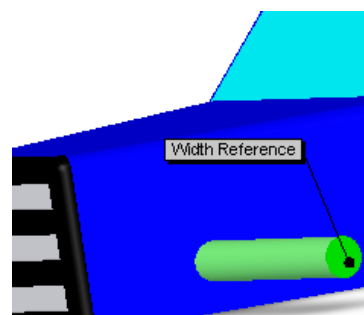
SolidWorks will automatically add  
A concentric mate to the axle and slot as  
Shown.



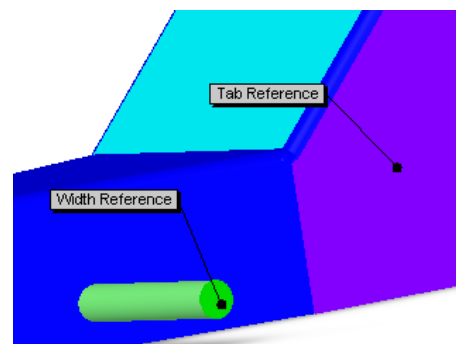
We will now add a width mate to the axle  
And car body. This time, however, we will  
Not be able to use the side surface at the  
Front of the car as it is set at an angle.

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 selection, as shown.

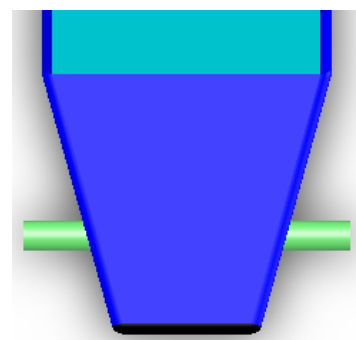


Next select the straight side surface  
Of the car body as shown.



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

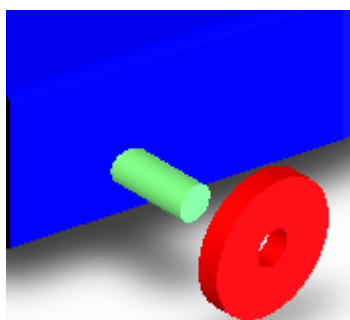
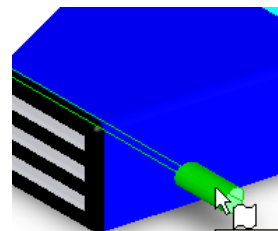
Selecting a “Top View” will allow  
You to see the position of the axle.



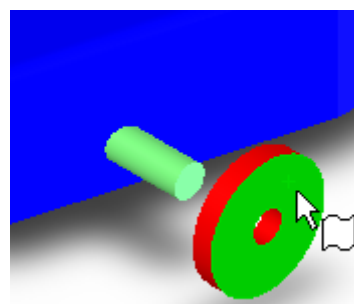
The next step is to mate the wheels to  
The axles.

Select the Mates icon from the assemblies  
Toolbar.

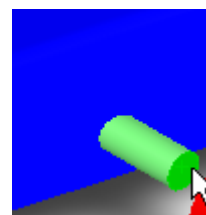
Next select the outside surface of the axle  
And the inside surface of the hole through  
The wheel and click OK. This will make the wheel  
and the axle concentric, as shown.



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

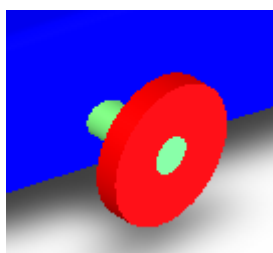


Now select the end surface of the  
Axle, as shown



Click OK  to finish mate.

This will align the two selected surfaces.



Repeat these steps for each of the remaining three wheels.



Finished Assembly:

