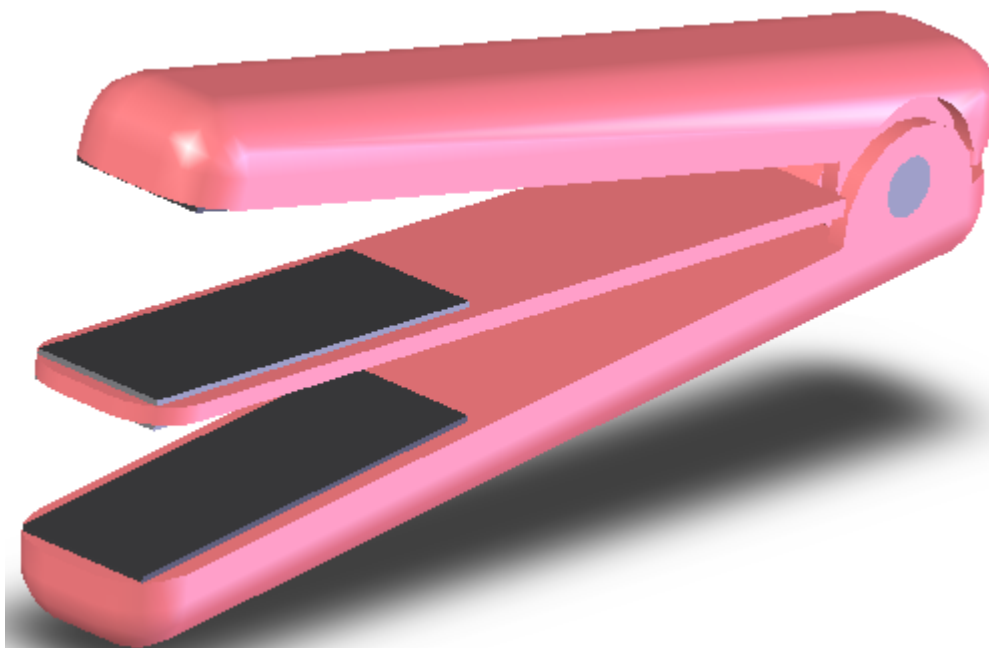


EXERCISE ONE:

GHD- HAIR STRAIGHTENER.

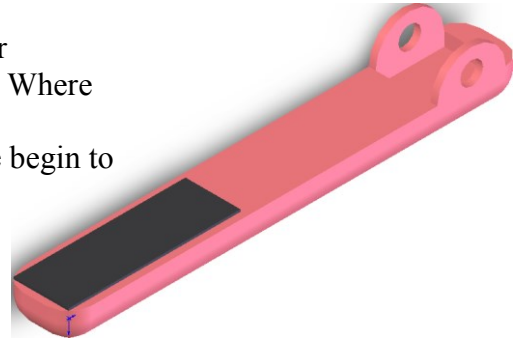


Prerequisite knowledge	Students should have completed Exercises from the file: <u>Introduction to Assemblies – Concept Mates and Beach Buggy Exe.</u> Students should also be familiar with Solidworks commands and the Solidworks interface.
Focus of lesson	This lesson will focus on using the following commands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

PART ONE: THE BASE

Look at the completed model, consider the best method of creating the model. Where should we begin?

Questioning the steps to use before we begin to create the model allows us to create it with the least number of steps it will allow us to modify it easily if necessary. This process is known as *“design intent”*.



Save File

Open a new part document and select **File, Save As**, name of file: **GHD Base**.

Getting Started

Select the Front Plane from the manager tree, as this is the plane that will allow us to create a sketch profile which creates most of the model.

Creating the Sketch:

On the Front plane, create the following sketch.

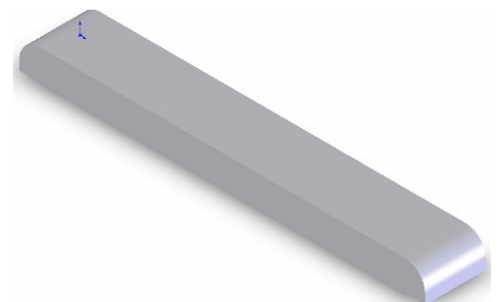


Note the position of the Origin in the bottom left hand corner. Only by relating the origin to our initial sketch can we fully define the sketch.

Creating Feature:

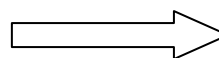
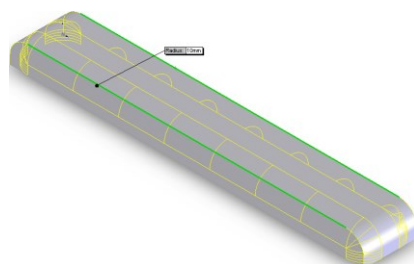
Extrude the sketch a depth of 30mm.

Rename the feature- “Bottom Case”-



Insert Fillet:

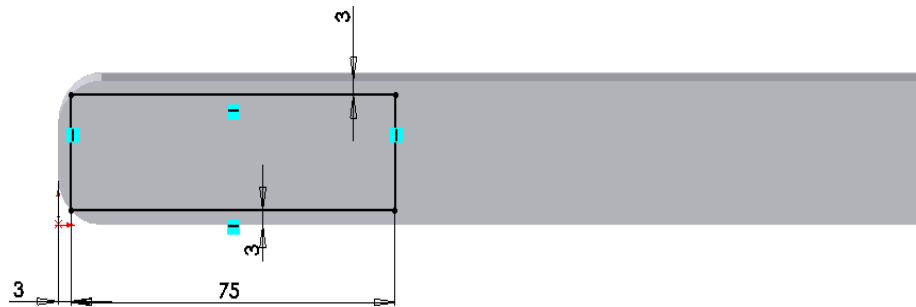
Insert a 10mm Fillet to the Top edges.



Rename the Fillet as “Corner Fillet”.

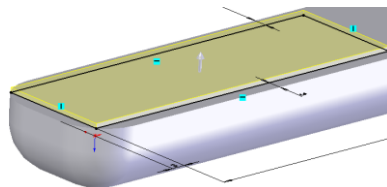
Creating the sketch:

Create the given sketch on the flat underside of the model.



Extrude Feature:

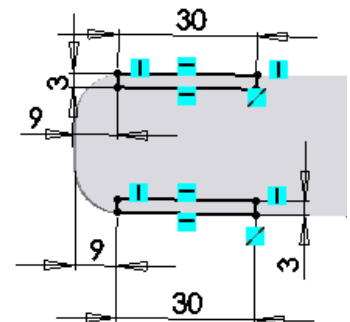
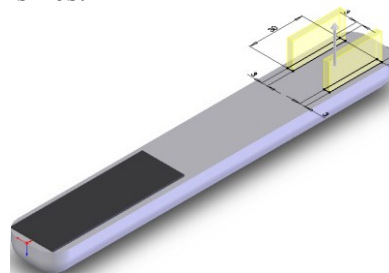
Extrude the sketch up to a height of 1mm as shown.



Rename the feature as “Heat Pad 1”.
apply colour as to heat pad.

Creating Sketch:

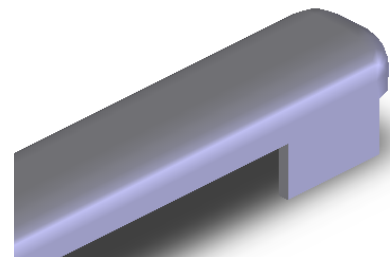
Select the face shown and create two sketches to the given sizes.



Extrude Feature:

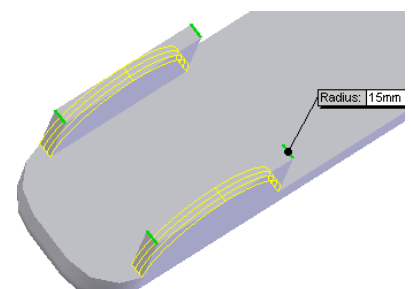
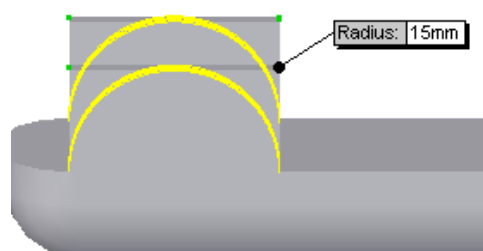
Extrude the sketches to a height of 15mm.

Rename as “Hinges”.



Fillet:-

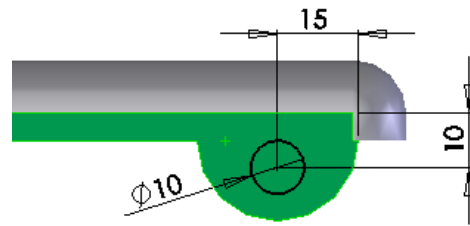
Select Fillet and create 15mm fillets on the hinges.



Rename as “Hinge Fillets”.

Create Sketch:

Create a sketch on the shown face to the dimensions given.

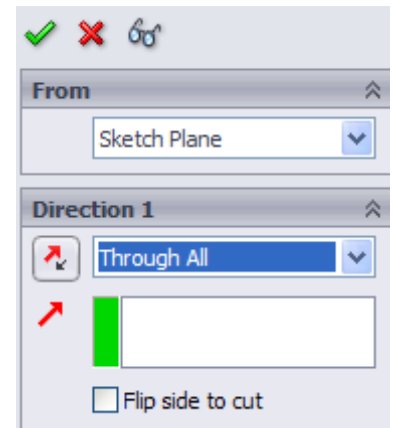


Feature Extrude Cut:

Select Extrude Cut from the features toolbar and select the “Through All” option.

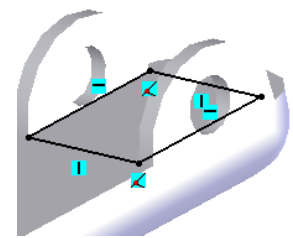
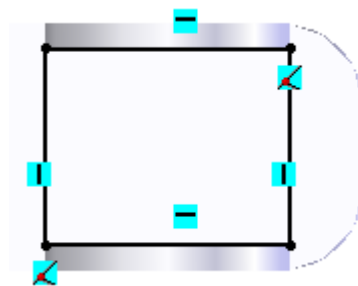
Rename feature as “Pin Hole 1”.

Note: *By using the Through all condition we can cut the hole through both of the hinges. This eliminates the need to repeat this step for the second hinge.*



Create Sketch:

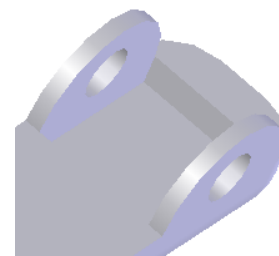
Create a sketch on the given face as shown. Note: the sketch is fully defined as it is coincident with the end points of both hinges.



Feature Extrude Cut:

Extrude cut to a depth of 7mm.

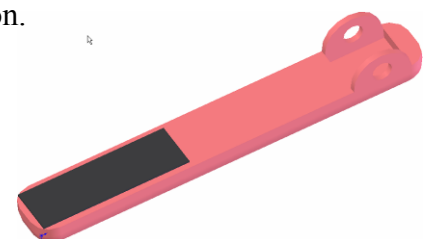
Rename as “Clearance Cut”



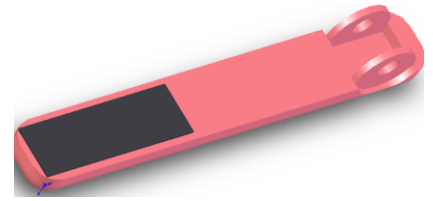
Apply Colour:

Apply colour as in previous exercises, to create Contrast between surfaces and add definition.

Close File.



PART TWO: THE TOP



Save File

Open a new part file, save file as : GHD Top.

Getting Started

What Profile would best allow us to create this model?, what plane would best allow us to create the profile?

Creating the Sketch:

To create this sketch use the front plane as it allows us to sketch the profile of the model and extrude it in the least amount of steps.

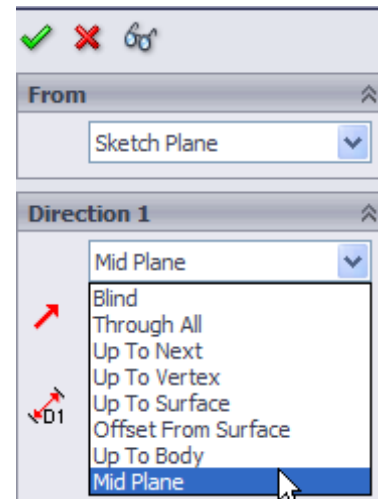
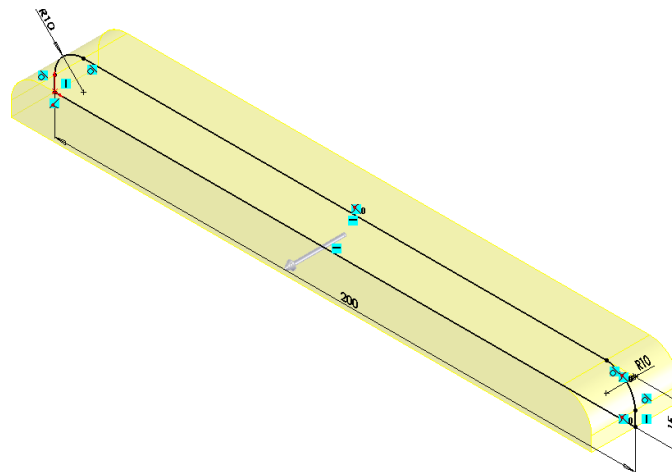
On the front plane create the following sketch.



Later we will want to mirror cuts from one side of the model to the other. To allow for this to be done easily we will extrude the model as a Mid Plane Extrusion. This will keep the front plane in the centre of the model.

Creating the Feature:

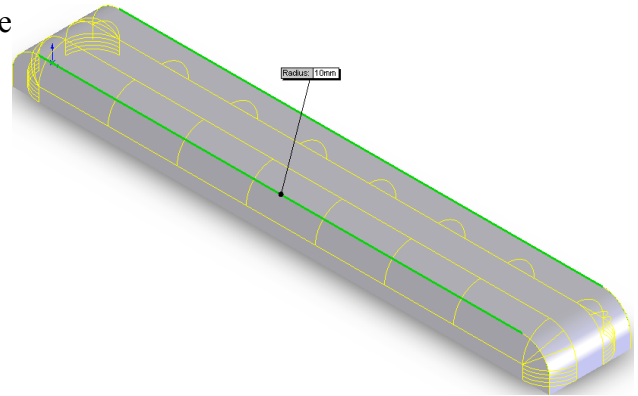
Extrude the sketch a depth of 30mm, using the Mid plane condition as shown below.



*Note: the position of the sketch in the centre of the model.
Rename the feature- "Top Case"*

Insert Fillet:

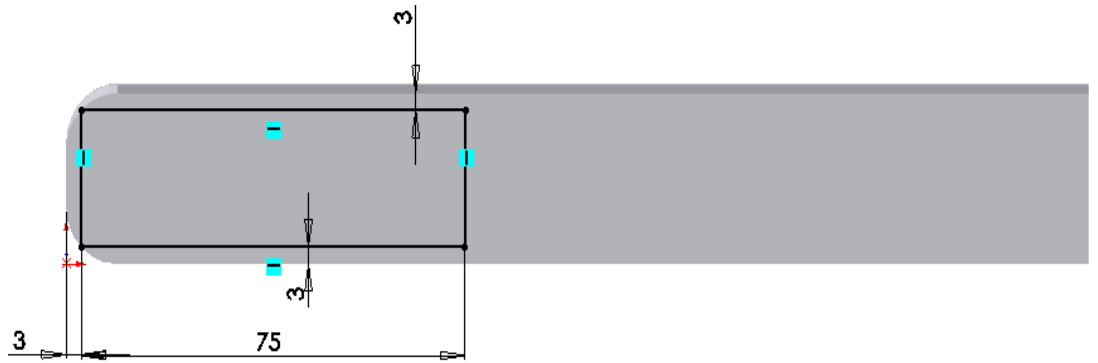
Create a 10mm Fillet to the top Side edges.



Rename the Fillet as “Corner Fillet Top”.

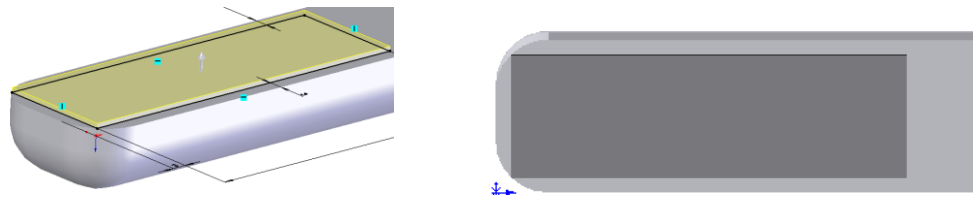
Creating the sketch:

Create the given sketch, on the flat underside surface.



Extrude Feature:

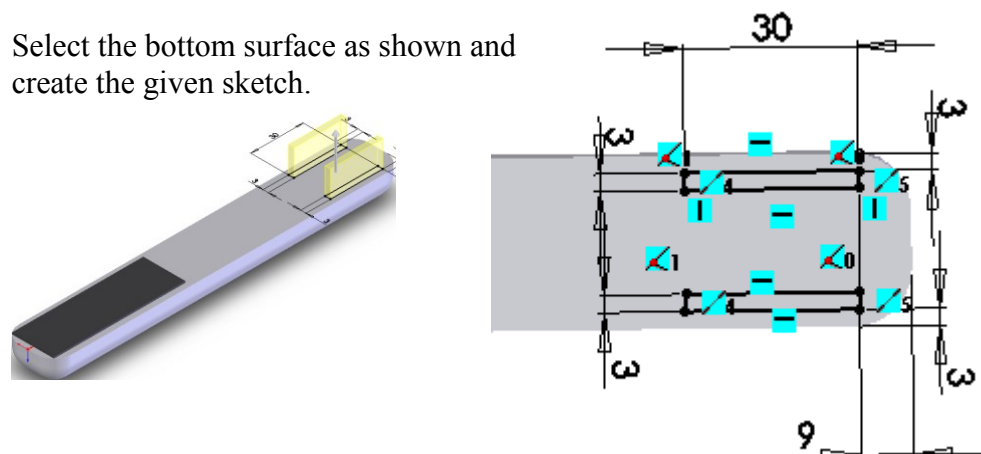
Extrude the sketch to a height of 1mm as shown.



Rename the feature as “Heat Pad 2”.

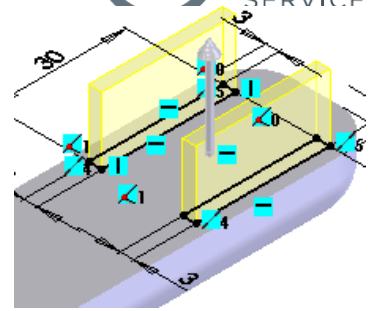
Creating the sketch:

Select the bottom surface as shown and create the given sketch.



Creating the Feature:

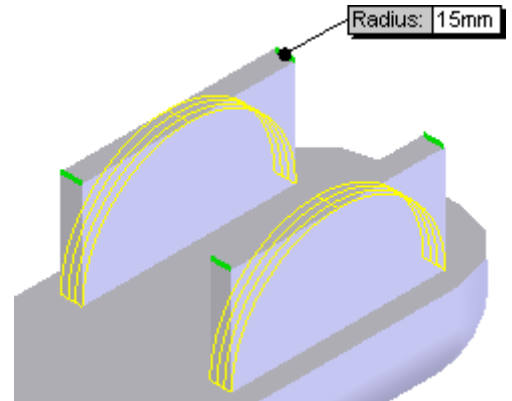
Extrude the feature to a height 15 mm.



Rename Feature “Top Hinge”.

Insert Fillets:

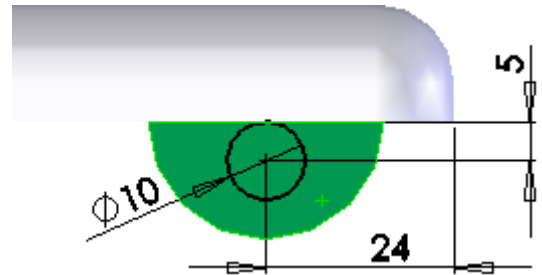
create a 15mm fillet on the hinges as shown.



Rename Feature “Hinge Fillet”.

Create Sketch:

Create a sketch on the given face to the dimensions shown.



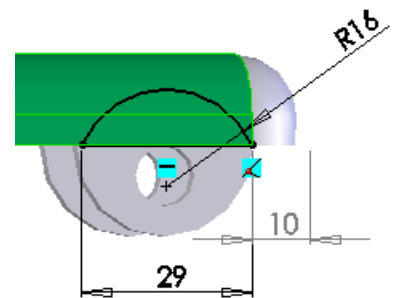
Feature Extrude Cut:

To cut the 10mm hole through both hinges, what option should be used from extrude cut?
we can use the Through all condition, which will cut the hole through both hinges.

Rename feature as “Pin Hole 2”.

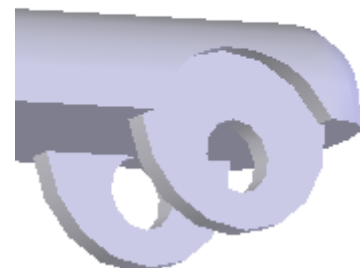
Create Sketch:

Create a sketch on the given face to the dimensions shown



Create the feature:

Extrude cut to a depth of 3mm.



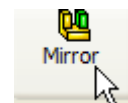
Rename “Hinge Clearance Cut”

Using Mirror:

The hinge clearance cut must be added to the other side of the model. To do this we could redraw the sketch on the other side but it is easier to mirror the hinge cut feature. Mirroring the feature also means that if a change needed to be made to the hinge cut, the change need only be made to one side. To add the same cut on the other side of the feature it is necessary to select a plane to mirror it about. As the front plane is in the centre of the model we can mirror the cut about it. This is only possible because the steps needed to create the model effectively were considered at the start.

Mirroring a Feature:

To mirror the “Hinge Clearance Cut” select mirror from the features toolbar

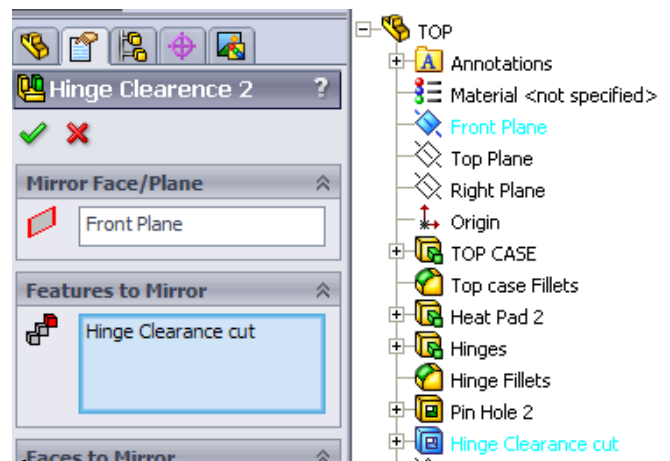


Next left click on the Front plane to select it and select “Hinge Clearance Cut” from the manager tree .

Select 

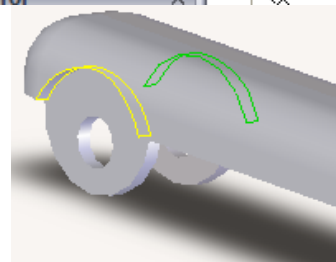
Note: Plane used →

Feature used →



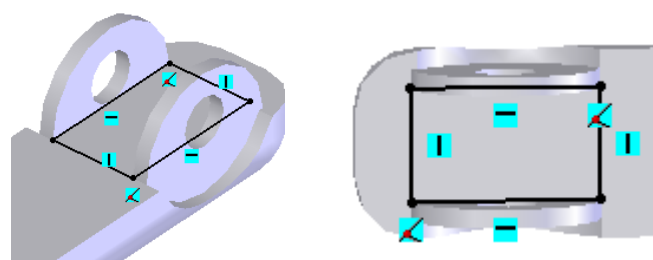
The cut will now be mirrored

Rename as Hinge Clearance 2.

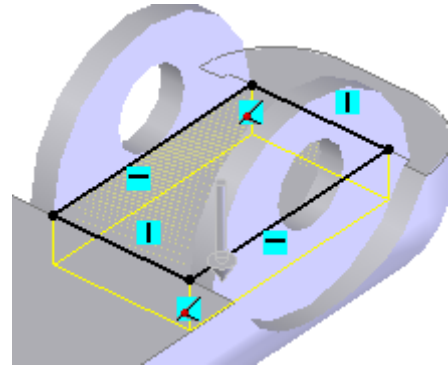


Create Sketch:

On the given surface create a Sketch as shown, ensuring that The sketch is coincident with the inside of both hinges, which are both fully defined.



Create Feature: Extrude cut to a depth of 7mm.
Rename Feature “Clearance cut”.



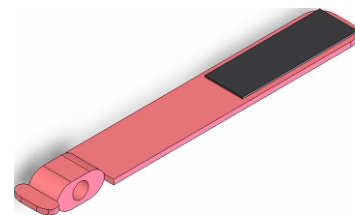
Apply Colour: Apply colour to match.

Close File.

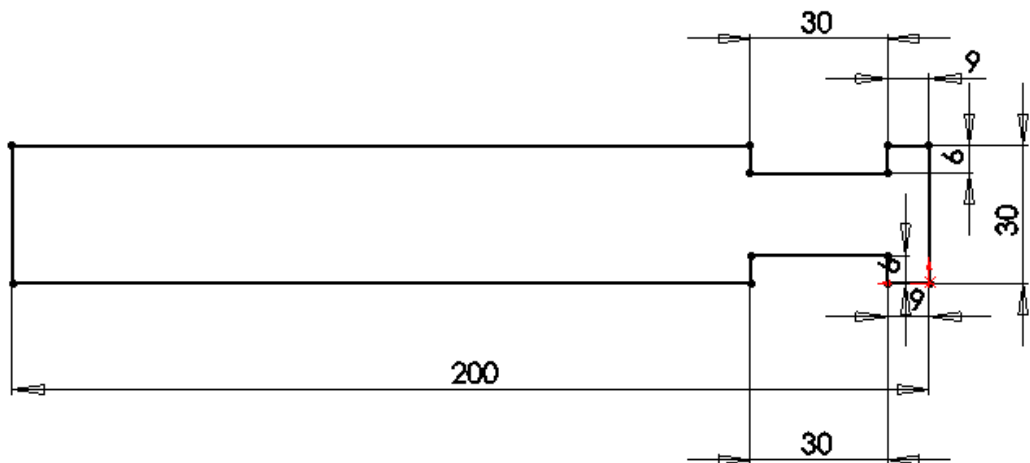


PART THREE: THE CENTRE.

Save File Create a new part file.
Save As - “Centre”



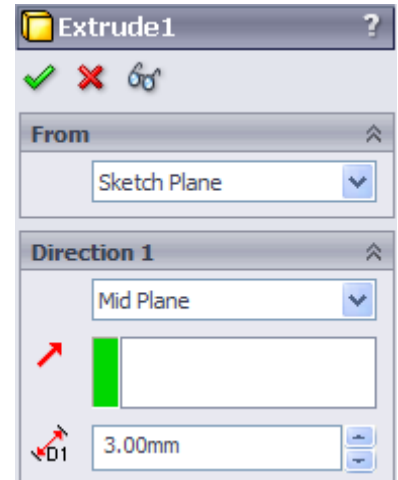
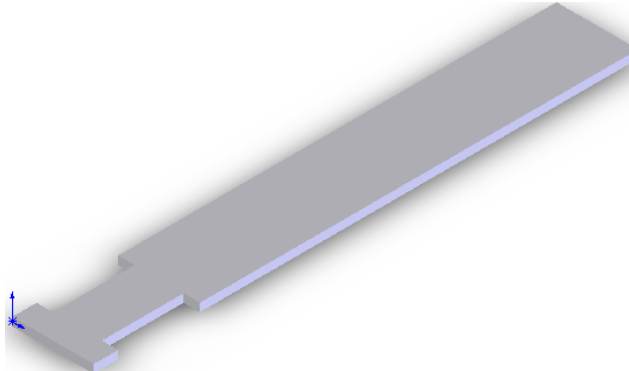
Getting started: On the top plane create the given sketch.



*Note; for clarity the Sketch relations have been turned off.
To turn the sketch relations on/off select “View” and “sketch relations”*

Creating a Feature:

Extrude the feature a height of 3mm using the Mid plane condition.



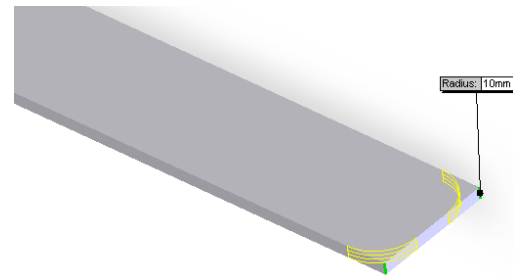
Note:

By using Mid plane the Top plane will be in the centre of the model, this will allow us to mirror the Heat pad from one side of the model to the other later.

Rename feature “Centre”.

Insert Fillet:

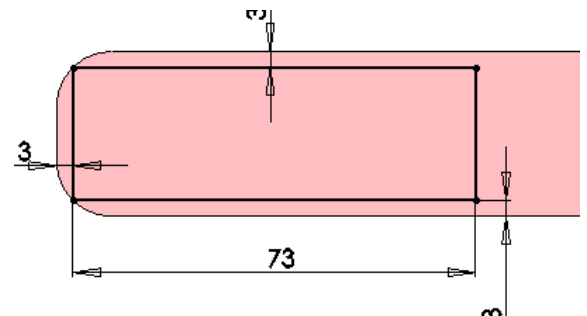
Insert a 10mm fillet on the front corners.



Rename “Corner Fillet”.

Creating Sketch:

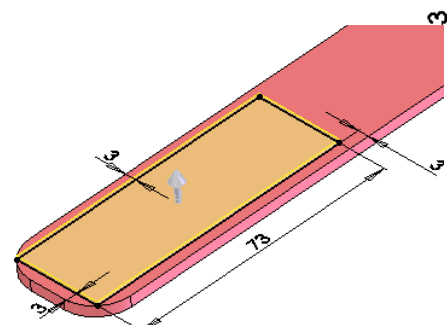
On the top surface create a sketch.



Creating Feature:

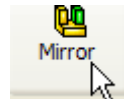
Extrude sketch to a height of 1mm.

Rename “Heat Pad 3”.

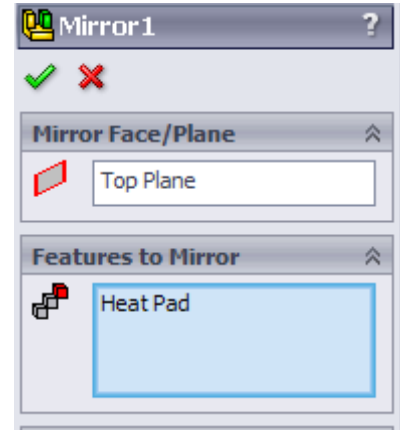


Mirror:

To mirror the “Heat Pad” select
From the features toolbar.



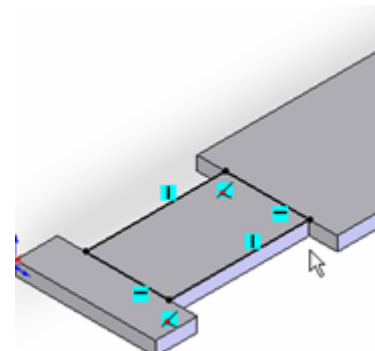
On the Mirror options box
Which appears on the left hand
Side select the “Top Plane”
And the “Heat Pad”.



Rename as “Heat Pad 4”.

Create Sketch:

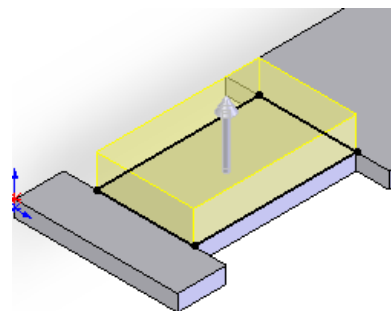
Create a sketch on the top surface as shown.
Note that no measurements are needed to fully
Define the sketch as it follows the contours of the
Extruded model which is fully defined.



Create feature:

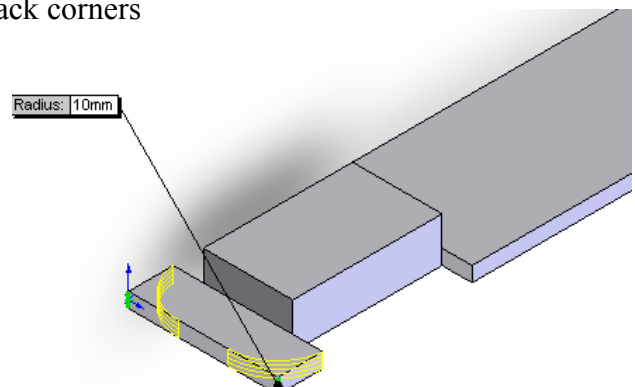
Extrude the sketch to a height of 6mm

Rename as “Pivot top”.



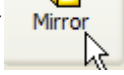
Insert Fillet:

Create a 10mm fillet at the back corners
As shown.

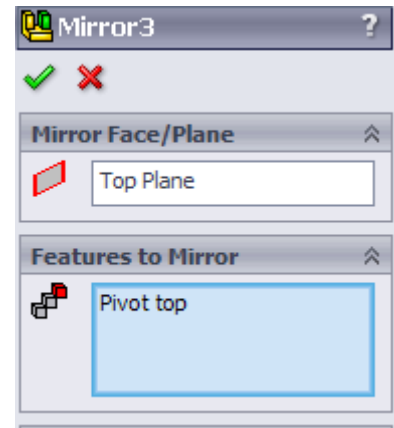


Rename “Back corner Fillets”

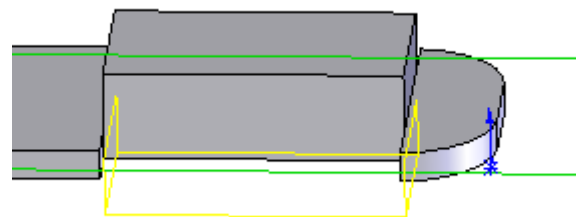
Mirror:

Next select Mirror  from the features toolbar .

On the Mirror options box
Which appears on the left hand
Side select the “Top Plane”
And the “Pivot Top”.

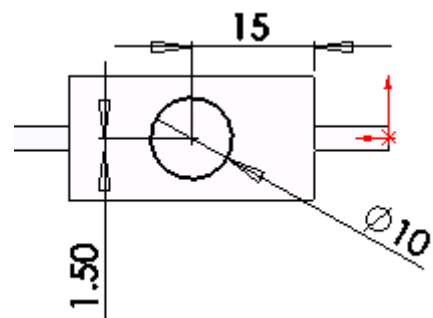


The Top Pivot will now be mirrored about the top plane.



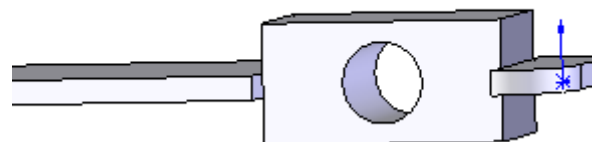
Create Sketch :

On the side surface of the
Pivot create the given sketch as
Shown.



Extrude Cut:

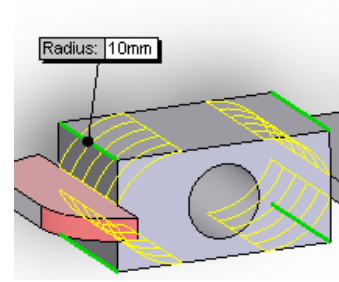
Extrude cut the Sketch by selecting the
“Through all” condition.



Rename as “Pivot Hole”.

Insert Fillet:

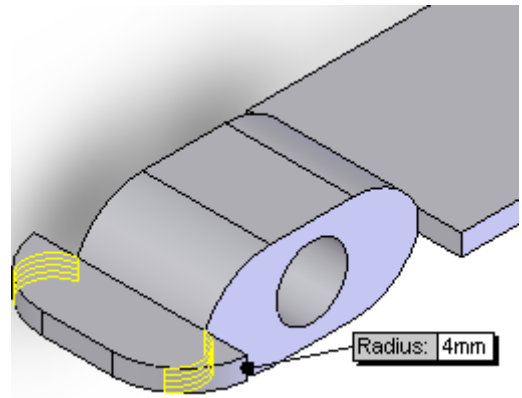
Create R10mm fillets to the four corners of the model as shown.



Rename as “Pivot Fillets”.

Insert Fillet:

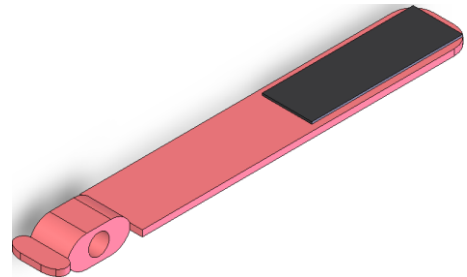
Insert R4mm to the back Corners.



Rename as “Back Fillets”.

Apply Colour:

Apply colour as in previous exercises.



Close File.

PART 4: THE PIN.

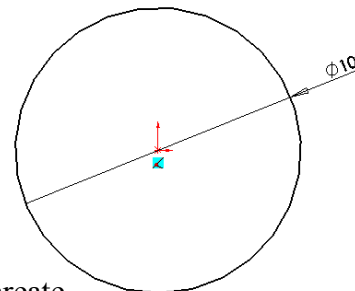
Save File

Select **File, Save As**, Filename “Pin”.

Create a sketch:

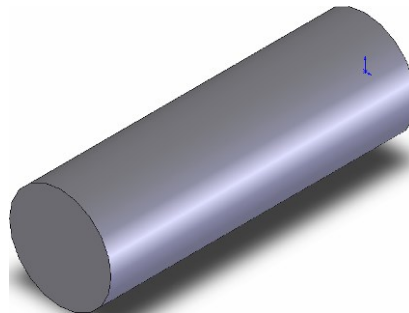
On the Front Plane create a sketch to the sizes shown.

The reason for creating the sketch on the front plane is that the pin will sit in the horizontal position in the final assembly, so it is best practice to create the pin in the horizontal position.



Create Feature:

Extrude the Sketch to
A height of 30mm.



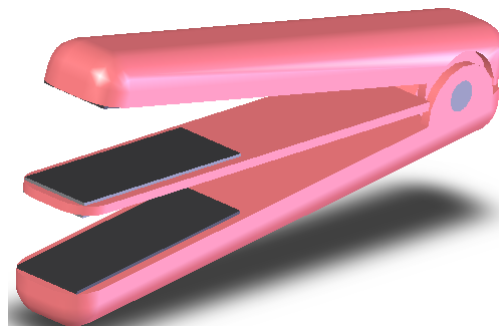
Apply Colour:

Apply colour as in previous exercises.

Close File.

ASSEMBLY.

(Students should have the introductory folder and the Beach Buggy assembly completed in Order to complete this assembly)



Save File:

Open an Assembly file, select **File, Save As,**
Filename “GHD”.

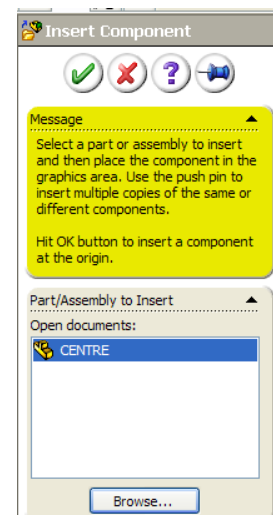
Insert Component:

The first component that is inserted will be fixed in place. With the GHD the Top and bottom need to be free to move so the first component to be brought in should be the centre.

Select Insert Component
from the assemblies toolbar.

Select browse and then select the centre model.


Select the  to position the Model on the origin.



Insert Component:

Select Insert component again
And insert the “Top”.

*Note: That when the centre is inserted an (f) will appear beside its name in the manager tree
This shows that it is fixed in position.*

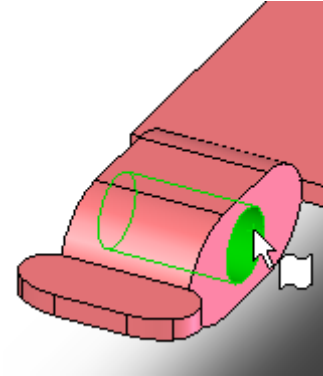
 (F) CENTRE<1>

Mating:

Select mate from the assemblies toolbar.

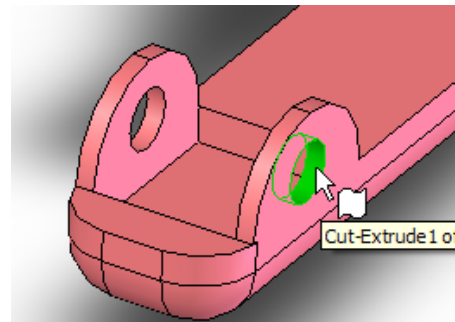
Surface selection:

To mate the Top to the Centre first select the inside of the hole in the Centre as shown.

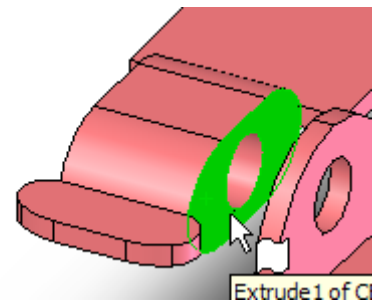


Select the inside surface of the axle hole from the base as shown. This will align the holes concentrically.

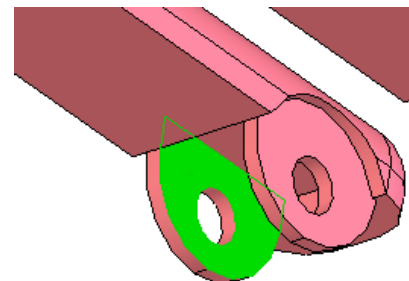
Click OK  to confirm.



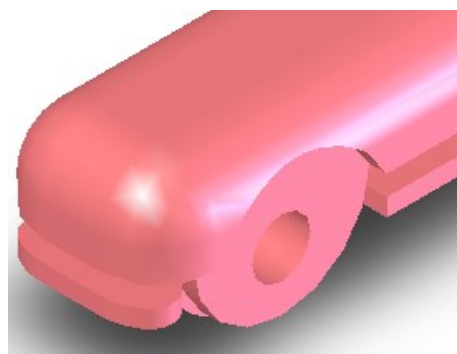
Select the outside face of the Pivot.



Select the inside surface of the Top pivot.



This will mate the selected surfaces.



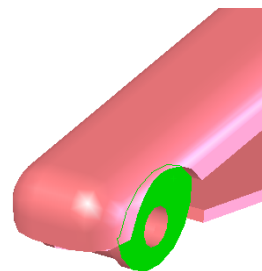
Insert Component: Insert GHD Base.

Mate: select mate and select the inside surfaces
Of the pin holes of the Base and
Top.

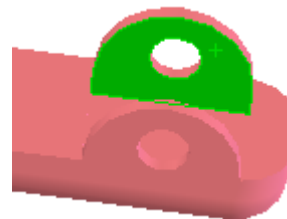


This will mate the holes concentrically.

Select Mate and now select
The outside face of the Top
As shown.



Next select the inside face
of the base.



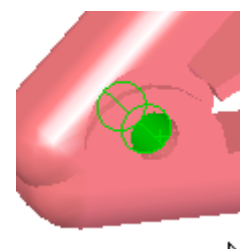
This will mate the selected
surfaces.

Insert Component: Next from Insert Components
insert the Pin.

Mate: select mate and then select the outside surface
of the pin.



Also select the inside surface of the pin hole,
This will make both surfaces concentric.



*Note: the Pin is mated concentrically with the
hole but is still free to move along the axis of the
hole. There is one more mate to complete before the
assembly is finished.*

Finally to finish fully mate the pin and base we must mate the pin flush with the base. How can we do this?

Select mate, next select the top surface of the Pin and the side surface of the GHD. This will mate both components flush.



Assembly Completed.

