

# **Steam Roller**





# **Object Analysis sheet**





Prerequisite KnowledgePrevious knowledge of the following commands is required to complete this<br/>lesson. Sketching (Line, Rectangle, Arc, Add Relations, Dimensioning),<br/>Extrude, Assemblies.

Commands UsedThis lesson includes Sketching (line, circle, arc, Smart Dimension), Cut Extrude<br/>with a line, Add relations, Sweep Boss/Base, Appearance, Assemblies, and<br/>Advanced Mates.

**Getting started** Create a **New Folder** in your chosen location called '**Steam roller**'.

The nine parts that make up the project will be saved here.

#### Part One – Steam Roller Body

Open New Part from the SolidWorks Document dialog box. Select File. Click Save as on the standard toolbar. Save as 'Steam roller body' in the 'Steam roller' folder. Continue to save periodically throughout the exercise.

**Create sketch** 

Create a sketch on the **Top Plane** using the dimensions shown.

Confirm the sketch.

**Extrude** the sketch to a **depth** of **63mm.** Use a **Blind** end condition

Rename the feature as 'Main Body'



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Sketching recess for Cab

In the sketch menu select **Rectangle** command and draw the rectangle shown. Add the dimensions shown to fully define the sketch.

Exit the sketch.

Select Extruded Cut to a depth of 20mm.

Rename the feature cut as 'Cab'

**Sketch for Engine** 

On the top surface draw the **rectangle** shown to the following dimensions.

Sketch a centerline on the top surface so that the rectangle can be mirrored.



Click the ok button voi to accept.

Confirm the sketch.

Select Extrude Cut and Through all.

Rename feature as 'Engine'.





TECHNOLOGY

SUBJECTS SUPPORT

SERVICE



CHNOLOGY Assembly Modelling JBJECTS SUPPORT Steam Roller Exercise SERVICE **Creating the Nose** Section Select sketch from the command manager. Choose the front face shown Center.... Arc draw the arc Using Centrepoint Arc to the dimension shown Exit the sketch and select Extrude Cut and **Through all**. Flip the side to cut if necessary. Flip Cut Rename the feature as 'Front curve'. **Creating Recess** for Upright Select sketch from the Command Manager and select the top face. Select Normal to 🔝 from the custom tool bar. Creating the sketch Sketch the rectangle shown to the given dimensions. Exit the sketch and select Extruded cut and Through all as the end condition. Rename the feature as 'Cut for upright'. **Creating the holes** for dowels On the surface shown draw a rectangle using the centerline command. Add the dimensions shown

**Design & Communication Graphics** 

**Add Relations** 

**Hole for Back** 

Axle



Exit the sketch and use Extruded cut through all. Rename feature as 'Hole for back axle'.

**Hole for Exhaust** Pipe

On the top surface sketch a circle to the dimensions given.

Draw a circle at each corner.

Exit the sketch.

dimensions.

Select **Extruded cut** to a distance of **10mm** 

Rename the feature as 'Hole for exhaust Pipe'.



#### **Edit material**



In the design tree right click on Steam roller body/ Appearance/ Edit material. Select 'Pine' from the 'Woods' folder and adjust the scale and angle of the grain.



**Design & Communication Graphics** 



Apply an **'Unfinished Pine Endgrain'**' texture to the faces that contain end grain as shown. Select **all** of the faces together and right click on Face<1>@Cab and select **Unfinished Pine Endgrain** from the Pine folder. Adjust the colour, scale and angle of the grain



#### Part two - Front Upright

Open **New Part** Save part as 'Front upright' in the 'Steam roller|' folder.

25 Sketch Sketch a rectangle on the Front Plane. Add dimensions. ഗ L Accept the sketch and extrude by 63mm. Rename the feature as 'Front upright'. **Drilling Hole** On the top surface draw a centerline between the midpoints of the opposite short edges. Draw a circle on this centerline and dimension as indicated. ф**6.5**0 Accept the sketch.

> Select **Extruded Cut** and **Through all**. Rename the feature as 'Hole for front axle'.

Edit Material

Sketch

Apply a Pine material to the part and add an 'Unfinished Pine Endgrain" Material to the faces containing end grain.



#### **Part Three – Front Axle**

Open a New Part Save the part as 'Front Axle' in the 'Steam Roller' folder

Sketch the following circle on the Front Plane.

Accept the sketch and extrude by 60mm Rename the feature as 'Axle'.

**Edit Material** Apply a Pine material to the part and add an 'Unfinished Pine Endgrain" Material to the faces containing end grain

Save the part

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**Part Four – Front Wheel** 

Open a New Part Save the part as 'Front Wheel' in the 'Steam Roller' folder.

Sketch

Sketch the circle shown on the Front Plane.

Accept the sketch.

Extrude the sketch by 50mm. Rename the feature as 'Wheel'.

Hole for Axle Sketch a circle on the face of the wheel. This circle must be concentric with the outer circle. Add dimension.

Accept the sketch.















#### Select Extruded Cut and apply a distance of 20mm.

#### **Edit Appearance**

Right click on the part in the design tree Select **Appearance** and choose black as the colour for the wheel.





Save the part

#### Part Five -Back Wheel

Open a **New Part** Save the part as 'Back Wheel' in the 'Steam Roller' folder.

Sketch the circle shown on the Front Plane.

Accept the sketch and **extrude** by **20mm**.

Rename the feature as 'Back wheel'..

Sketch another circle with diameter **9mm** on the face of the wheel as shown.

Accept the sketch and select **Extruded cut** With **Through all** as the end condition.





**Edit Appearance** 

Hole for Axle

Apply black as the colour for the wheel as before.



Save the part



#### Part Six – Back Axle

Open a **New Part** Save the part as 'Back axle' in the 'Steam Roller' folder.

SketchSketch a circle on the Front Plane with a<br/>diameter of 9mm.<br/>Accept the sketch.<br/>Extrude the sketch by 120mm.

Rename the feature as 'Back Axle'.

Edit MaterialApply a Pine material to the part and add an 'Unfinished Pine Endgrain"Material to the faces containing end grain



Part Seven – Cab Support

Open a **New Part** Save the part as 'Cab Support' in the 'Steam Roller' folder.

SketchSketch a circle of diameter 9mm on the Top Plane.<br/>Accept sketch.<br/>Extrude the sketch by 95mm.<br/>Rename extrusion as 'Cab support'.



Edit MaterialApply a Pine material to the part and add an 'Unfinished Pine Endgrain'Material to the faces containing end grain



Save the part

Sketch



#### Part Eight – Cab Cover

Open a New Part Save the part as 'Cab Cover' in the 'Steam Roller' folder.

Sketch a rectangle on the Top Plane. Dimension as shown.

> Accept the sketch and extrude by 10mm. Rename the feature as 'Cab Cover'.

Select the underside of the cab. Select **Normal to** 4 and draw the centerlines shown.

Sketch a circle at each corner.

**Add Relations** Use Add relations to make all the circles equal.

Dimension the diameter of the circles 9mm.

Accept the sketch.

Select Extruded cut and apply a depth of 6mm. Rename the cut feature as 'holes'.



Chamfer

Select **chamfer** in the features toolbar. Apply a chamfer of **5mm** to the top edges.



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Edit Material	Apply a Pine material to the part and add an <b>'Unfinished Pine Endgrain"</b> Material to the faces containing end grain			
	Save the part.			
<u> Part Nine – Exhaust Pipe</u>				
	Open a <b>New Part.</b> Save the part as 'Exhaust Pipe' in the 'Steam Roller' folder. To create the Exhaust pipe <b>Sweep boss/base</b> will be			
	<ul> <li>Is create the Exhibits pipe Sweep Sobs/Sube Boss/B will be Boss/B</li> <li>Is the sweep path.</li> <li>The sweep section.</li> </ul>			
Sketch 1.	Use the Line command on the Front Plane to sketch the profile shown. One end of the line segment is placed at the origin.			
Add Relations	Use Add Relations to make the end of the arc vertical with its center point			
	Add the dimensions. Confirm the sketch.			
Sketch 2.	Select sketch command again and select the <b>Top Plane</b> Exhaust pipe from the design tree.			
	Select Normal to Look perpendicular to the plane.			
	Draw the circle with its center point at the origin as shown.			
	Dimension the diameter <b>3mm</b> .			
	Design & Communication Graphics			



Confirm the sketch.

Sweep

Choose **Sweep Boss/Base** from the features toolbar. Select the circle as the profile and the line as the path.

Sweep ?				
✓ ×				
Profile and Path *				
Ś	Sketch2			
<del>ا</del> لا	Sketch1			
<u>O</u> ptions		*		
Guide <u>C</u> urves		*		
Start/End <u>T</u> angency ×				
Thin Feature ×				



Confirm the sketch. Rename the feature as 'Exhaust pipe'.

#### **Edit Material**

Select the material **brushed brass** from the brass folder as shown





Save the part.





### **Steam Roller Assembly**

Open an existing part	The part files for this assembly are saved in the f titled 'Steam Roller'. Open the part called 'Steam roller body'. Click <b>Make Assembly from Part/Assembly</b> .	Solider SolidWorks Fe Edit View Insert Tools Window Sketch Smart Dimension Features Sketch Make Drawing from Part Make Assembly from Part Begin Assembly ?
Save	<ul> <li>Insert component dialog box appears with 'Steam roller body' displayed.</li> <li>Click on in the property manager.</li> <li>The part origin will snap to the origin of the assembly.</li> <li>Select File, Save as on the standard toolbar. Save the assembly as 'Steam Roller' in the same</li> </ul>	Image: Constraint of the second se
Adding Component	Select Insert component Insert Compo from the Assembly toolbar. Choose Browse from the Insert Component dialog box. Click on 'Front upright', open and click in the graphics area to place the part as shown.	
Insert Mates	Use the <b>Rotate</b> components to rotate the components to the required orientation. The component of the required orientation. Select <b>Mate</b> from the assembly toolbar. Mate the back edge of the upright with the back of the trench shown.	part



A Coincident Mate will be selected by default.

Accept	<b>V</b> .
recept	<u> </u>

Mate the side of the upright with the side of the trench.

Accept 🖌

Finally mate the top of the main body with the top edge of the upright.



manager.



Tab



The upright is now fully defined i.e. it has no degree of movement

Inserting Further Components	Choose <b>Insert Component</b> from the assembly to Browse, select the 'Front axle' and insert into the graphics area as before.	olbar.
Insert Mates	Mate the surface of the dowel with inside surface of the hole	
	A Concentric Mate will be chosen by default.	
	Accept this mate.	Nate
Additional Mates	The axle has to be centered in relation to the upright	Mate Selections
	In the mate property manager expand <b>Advanced Mates</b> and select <b>Width Mate.</b>	Standard Mates        Advanced Mates        Ø Symmetric        Ø Cam        Ø Width
		Width3
	Select the two end surfaces of the axle for the <b>Width Selection.</b>	Mate Selections Width Face<1>@Part3_Front_a Face<2>@Part3_Front_a

Choose the two sides of the upright

**Adding Back Axle** 

**Insert Mates** 



for the Tab Selections.

Should any dimensions change the axle will always remain centered on the upright.

Accept the mate.

Select **OK** again to exit the property manager.



The axle is under defined i.e. it is free to rotate as it would in reality.

Select Insert Component from the assembly toolbar Choose the "Back axle" and drop it into position.

Follow the same process as for the front axle. Mate the surface of the axle with the inside surface of the hole.



Select Advanced Mates to centre the axle in Relation to the body of the steam roller.

Select Width in the Advanced Mates dialog box.



Width Reference

Select the two end surfaces of the axle as the **Width Selections** Select the two sides of the body as the Tab Selections





Select **OK** again to exit the property manager.



The axle is under defined i.e. it is free to rotate as it would in reality.





Tab Reference

**Design & Communication Graphics** 



Adding the Back Wheels

**Insert Mates** 

Select **Insert Component** from the assembly toolbar. Choose the "**Back Wheel**" and drop it into position.

Select **Mate** and select the surface of the axle and the inside of the hole.

A Concentric Mate is selected by default.



Mate the outside surface of the wheel with the end surface of the axle.

Accept 🥑 .

Select **OK** again to exit the property manager.

Follow this process again to assemble the other back wheel.







Inserting the Front Wheel

**Insert Mates** 

Select **Insert Component** from the assembly toolbar. Choose the "**Front Wheel**" and drop it into position.

Select **Mate** and mate the surface of the front axle with the inside surface of the hole.





Mate the end of the axle with the bottom surface of the hole



To select the bottom of the hole **Hidden lines visible** may need to be selected.

Accept 🖉.

Select **OK** again to exit the property manager.

Carry out the same procedure to assemble the other back wheel.



The wheels are under defined i.e. they are free to rotate as they would in reality.

Inserting the Cab Supports

Select **Insert Component** from the assembly toolbar. Choose the 'Cab Support' and drag it into position.

**Insert Mates** 

Choose **Mate**, select the surface of the support and the surface of the hole as shown.

Accept the Concentric Mate.

Mate the bottom of the hole with the end surface of the support.

Accept the Coincident Mate.







Mate the **Front Plane** of the assembly with the **Front Plane** of the 'Cab Support'. Apply a **Parrallel** mate.



Design & Communication Graphics

The component is now fully defined.

Select **OK** again to exit the property manager.





# Adding the other Supports

To add in the other **Supports** hold down the **Ctrl key**, select and drag the supports from the **Feature Manager Tree.** Drop them into the drawing area.

Mate the supports as before



**Inserting Cab Cover** 

Select **Insert Component** from the assembly toolbar. Choose the "**Cab Cover**" and drop it into position.

**Insert Mates** 

Select **Mate** and select the side of the cab cover and the side of the body. Apply a **Parallel Mate.** 



Mate the surface of the support shown with the appropriate hole.



Mate the end of the support with the bottom of the hole.

Accept 🧭

The component is now fully defined.

Select **OK** again to exit the property manager.







Inserting the Exhaust Pipe

**Insert Mates** 

Select **Insert Component** from the assembly toolbar. Choose the "**Exhaust Pipe**" and drag it into position.

Mate the surface of the pipe with the surface of the hole.

A Concentric Mate is chosen by default.

Accept 🥑.

Mate the bottom of the hole with the end of the pipe.

A Coincident Mate is chosen by default.

Accept 🖉.



#### Preventing the Pipe From rotating

To fully define the exhaust pipe, a parallel mate must be added between the **Right Plane** of the assembly and the **Front Plane** of the component.

Select Mate. Expand the "Exhaust Pipe" part in the graphics area. Select the **Front Plane** as shown.

Choose the Right plane of the assembly



Select **OK** again to exit the property manager.









# **Exercise Complete!**





## **Other Possible Assembly Options**