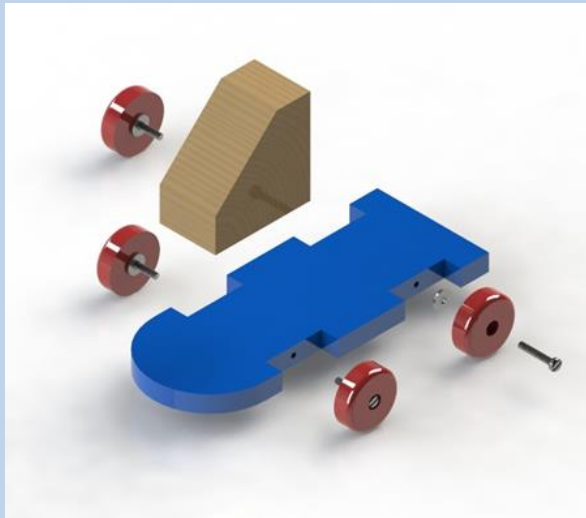![PDST logo] Professional Development Service for Teachers | An tSeirbhís um Fhorbairt Ghairmiúil do Mhúinteoirí

Introduction to SolidWorks for Technology
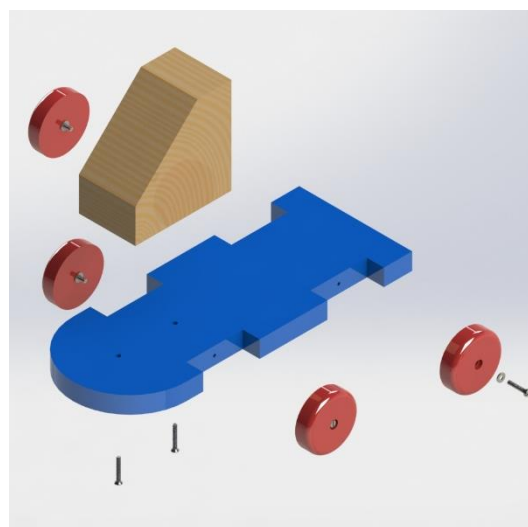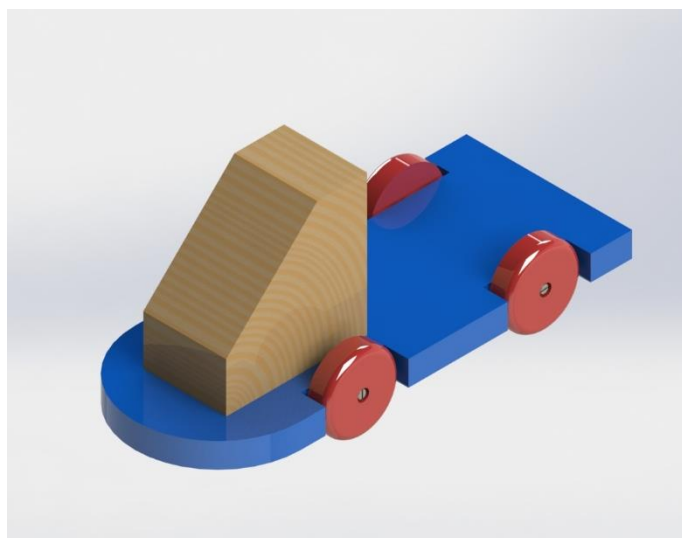
No1: Childs Toy

Table of Contents

Introduction

This exercise is an introduction to SolidWorks for Technology. The exercise looks at a technology project from all areas of SolidWorks; part modelling, assembly, drawings and photoview

Learning Intentions

At the end of this workshop you should be able to:

1. Create new **Parts** using basic sketching and feature commands
2. Create a new **Assembly**, mating two components and adding appearances and scenes
4. Complete **Drawings** of an assembly including a worksheet for students
3. Create and save photorealistic images using **Photoview 360**

Prerequisite Knowledge

None

Saving Your Work

Create Folder called **Childs Toy**. Save all files into this folder

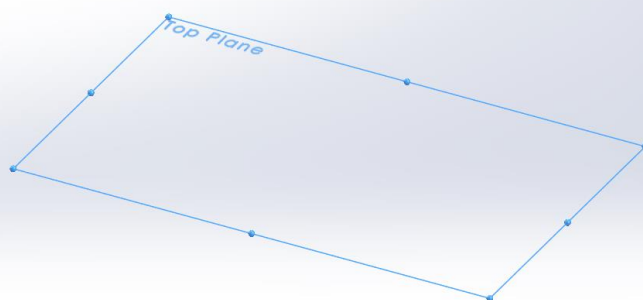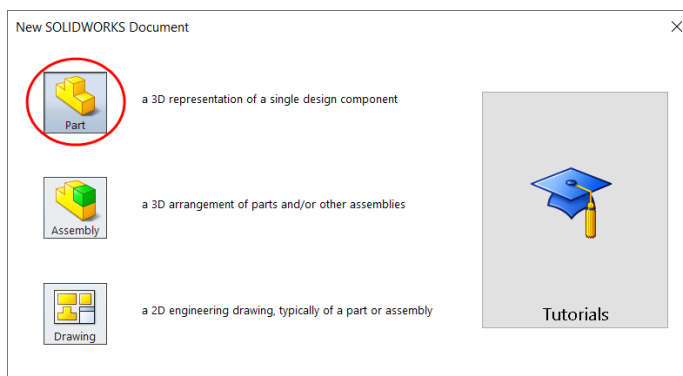# Part Modelling: Cab

Open a new part.
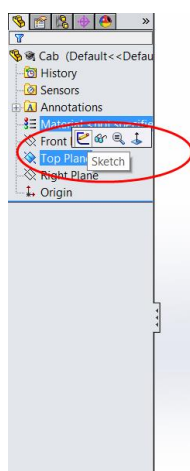
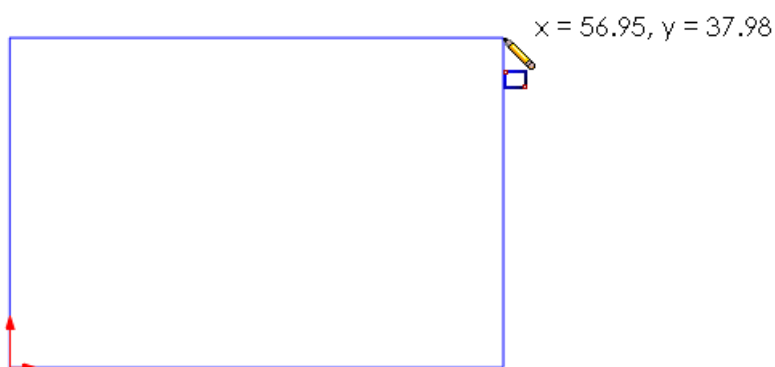Save Part as **Cab.**

Create a new

sketch

on the Top

Plane

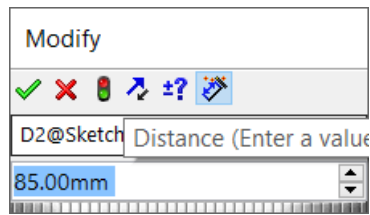Select the Rectangle command and draw a rectangle with one corner co-incident to

the origin. . Hold down the

left mosue button to drag out

rectangle, release when its an

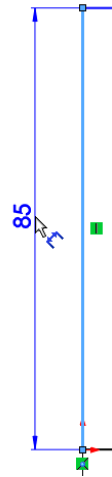approximate size. Accept the sketch

to complete .

$x = 56.95, y = 37.98$
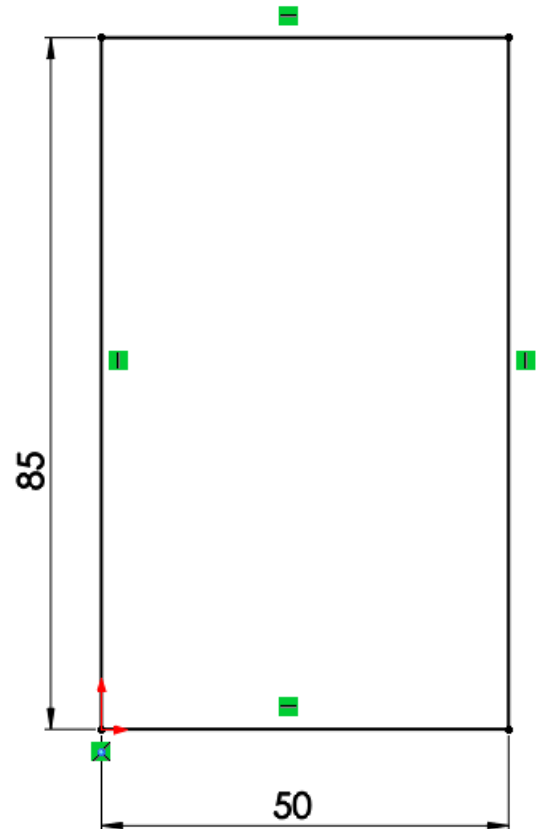
Select **Smart Dimension**

 and apply the

dimensions below to the

rectangle.

Hover over line, it will highlight in orange.

Left-click on line drag out dimension line

and left-click again to open the dimension

window.

The rectangle now turns black as it is **Fully**

Defined sketch.

Modify

D2@Sketch | Distance (Enter a value)

85.00mm

85

85

50

Boss-Extrude

From

Sketch Plane

Direction 1

Blind

85.00mm

Draft outward

Direction 2

Thin Feature

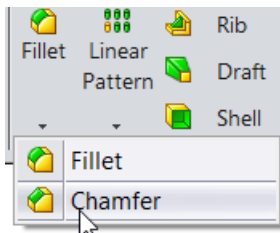Selected Contours

*Trimetric

 Features Select the Features Tab and select
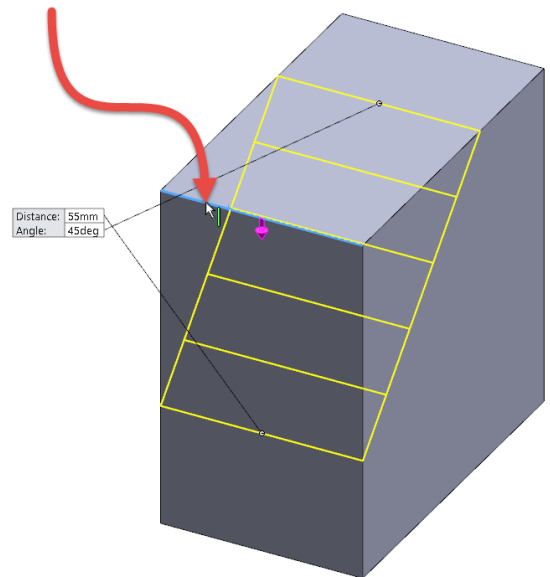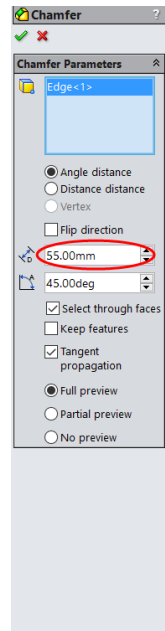
Extruded Boss/Base

The Sketch will move to an isometric view.

Add **85mm** dimension. Accept the extrude.

 .
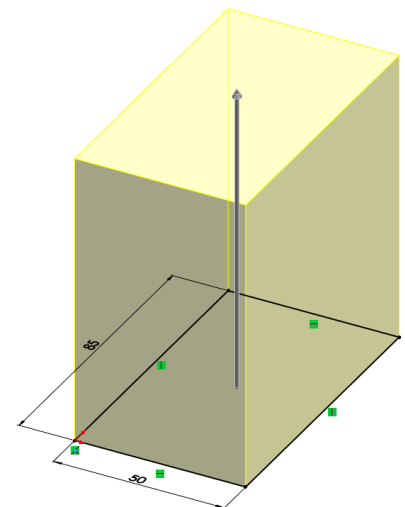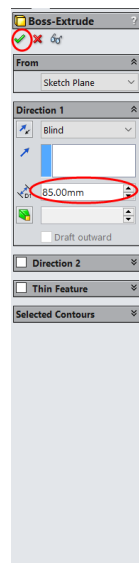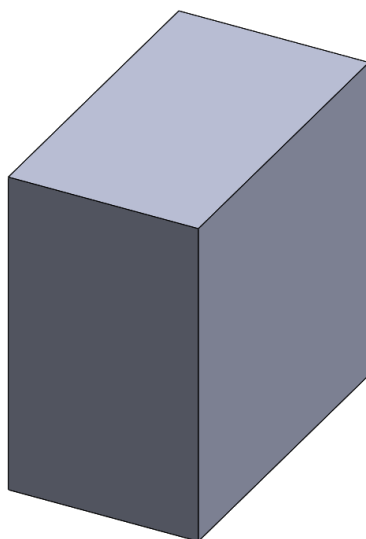
Select the Chamfer command. It is located under the Fillet Icon.

Apply a 55mm chamfer to the top edge as shown. The Full preview button will allow you to see the resulting feature. Click OK ✓.
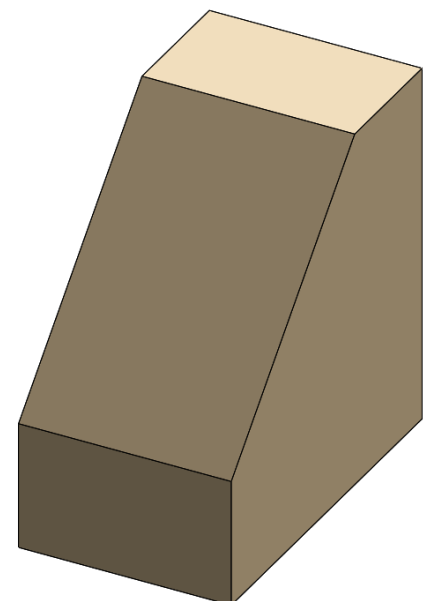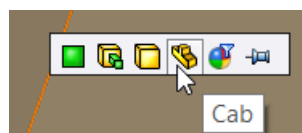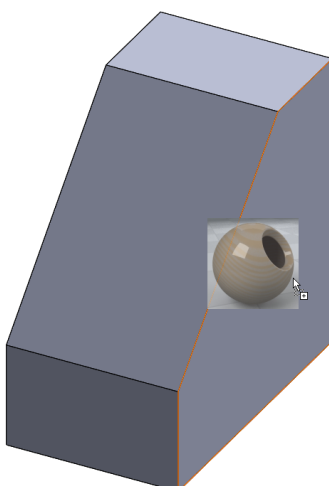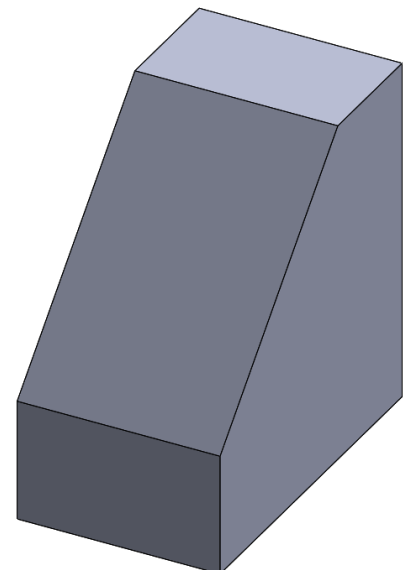Save your work.

Apply an appearance to the Cab. Select the **Appearance** tab on the right. Browse for

Organic, Wood, Pine. Select Polished Pine and drag and drop onto the Cab. A pop-up window will appear. Select the part icon to apply the appearance at part level. Save you work.
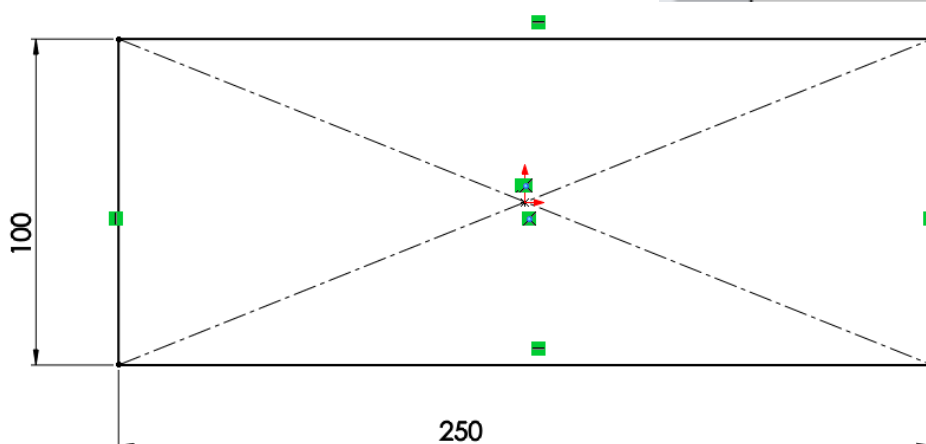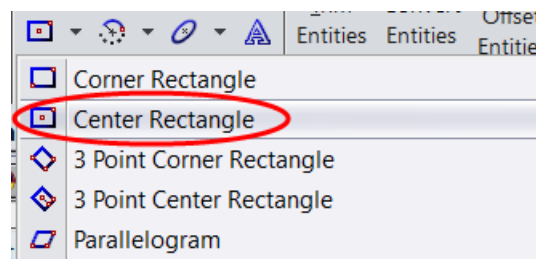
The Cab part is now complete

Part Modelling: Base

Base shape

Open new **Part** Part . Save as **Base** 💾 ▾. Create a sketch on the **Top Plane**. Draw a Corner Rectangle and dimension as shown.

- Corner Rectangle
- Center Rectangle
- 3 Point Corner Rectangle
- 3 Point Center Rectangle
- Parallelogram

100

250

Select **Extrude Boss/base** and extrude **15mm** with a blind end condition. Accept the extrude ✅
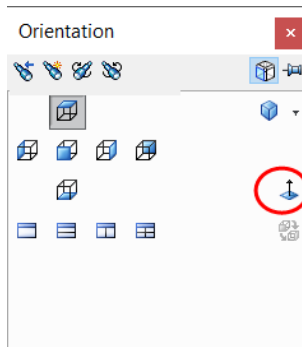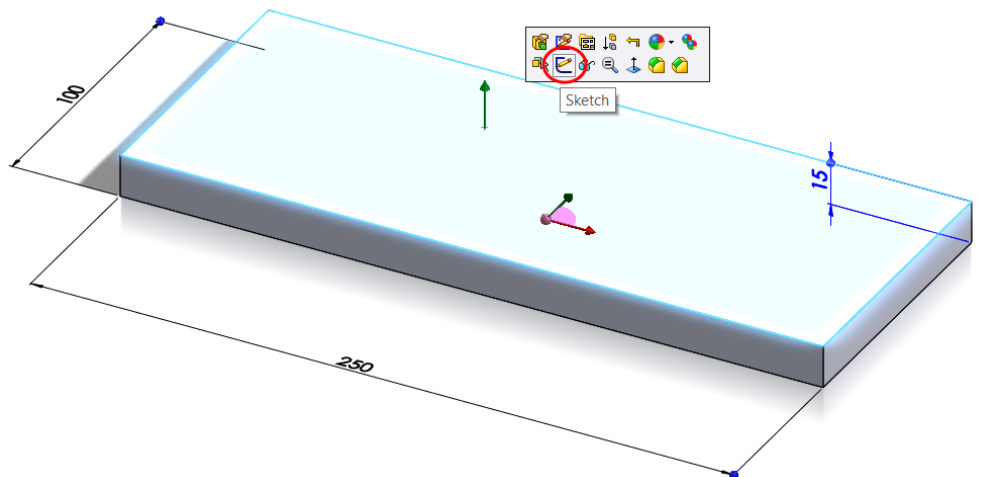
Rename the feature as **Base shape** (press Fn+F2)
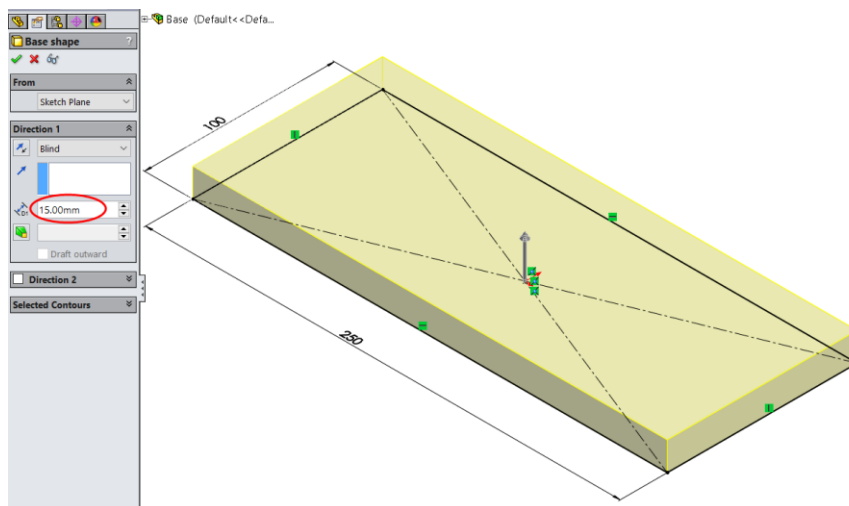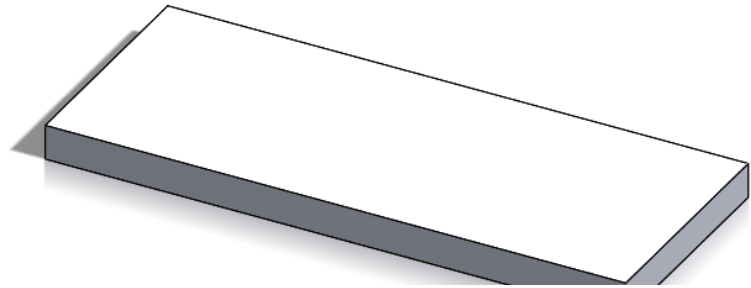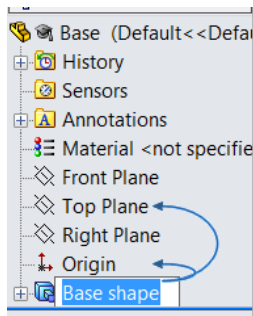
Wheel Notches

Create a new **sketch** on the top surface by left-clicking on the surface and selecting the sketch icon from the pop-up toolbar.
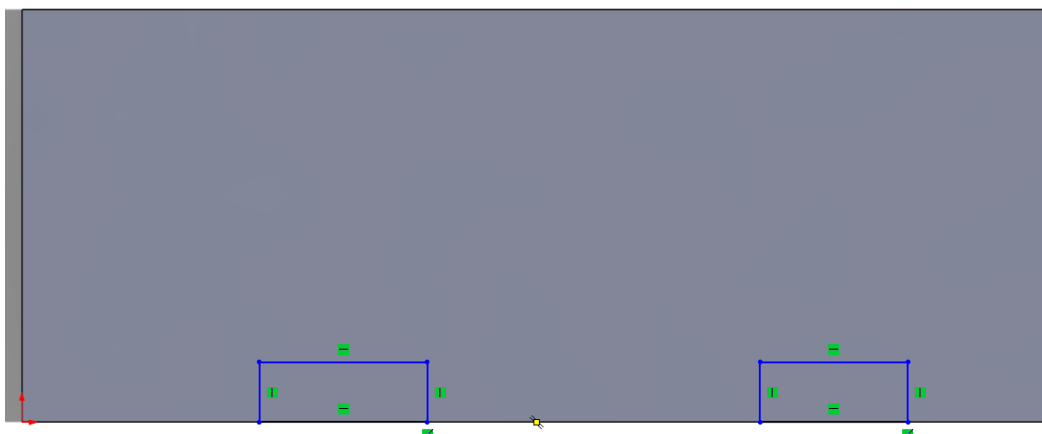

Select a normal-to view by pressing the space bar and selecting **Normal To**

Sketch two rectangles on the edge of the base as shown. The 1st corner should have a coincident relationship with the edge of the base, as shown by the cursor having a yellow coincident relation highlighted when sketching
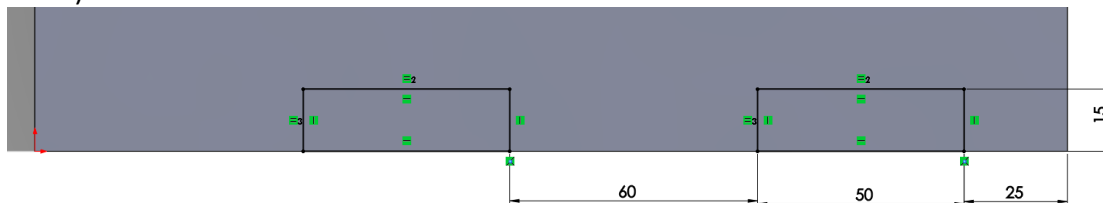

Add an **Equal** relation to the horizontal lines (hold Ctrl key and select lines). The pop-up toolbar will allow you to add relations.
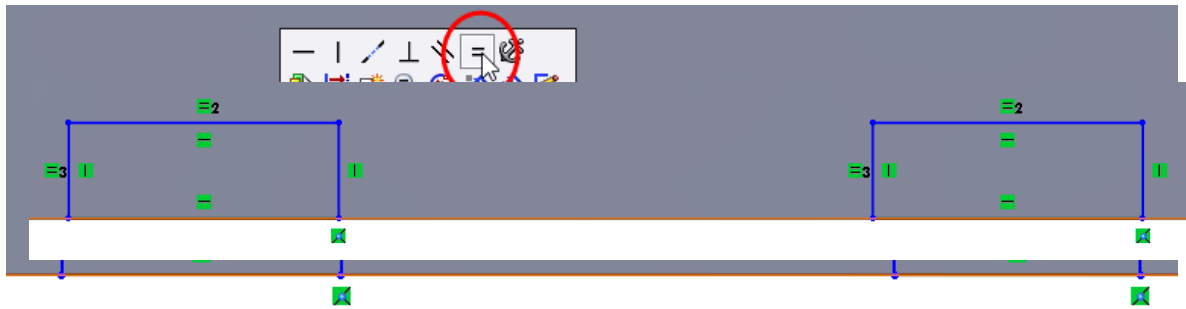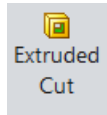

Repeat the add relation for the vertical lines. The equal relations can now be seen on the lines as shown

Smart Dimension ![Smart Dimension icon] the sketch as shown. The sketch is now full defined (sketch is now black).
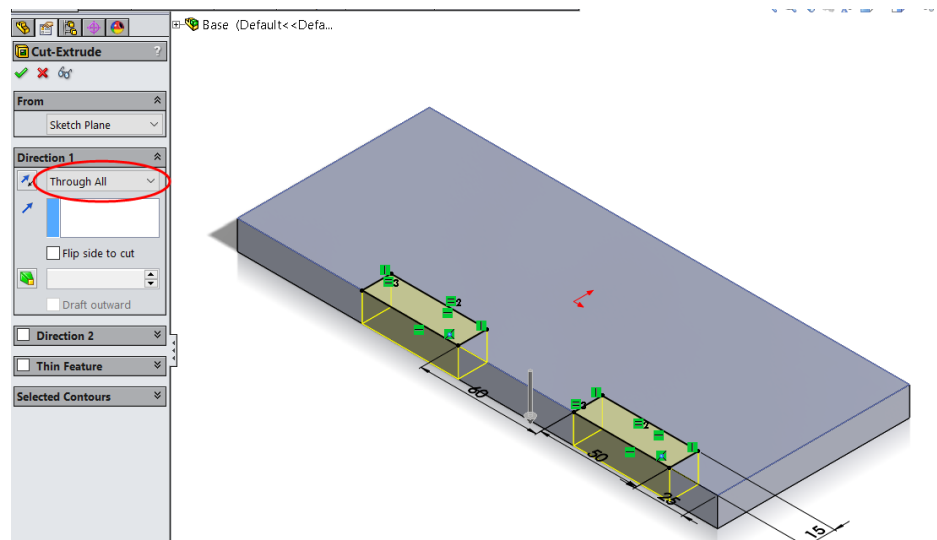

Select **Extruded Cut**

![Extruded Cut icon] from the
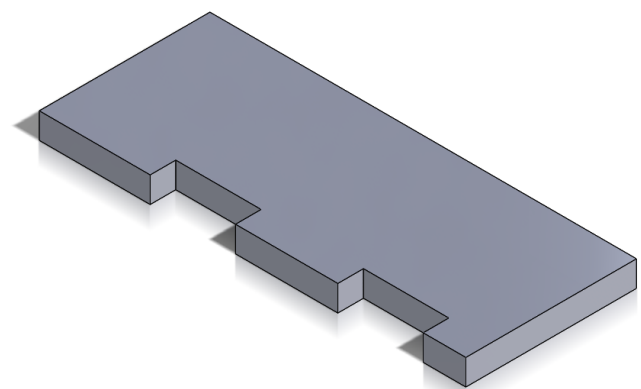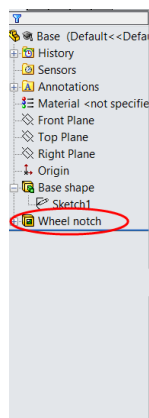
features tab and

change the direction to

Through All

Rename feature as
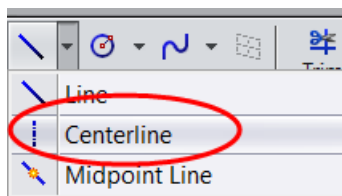
Wheel notch (ctrl+ F2)**.**

Click OK ✅

Save your work.

Tapped Holes for Wheels

Create a new **Sketch** on the vertical surface of the notch as shown.

Draw a **Centreline** diagonally across the surface.

Add a point [*] to the midpoint of the centreline. Repeat this for the other notch
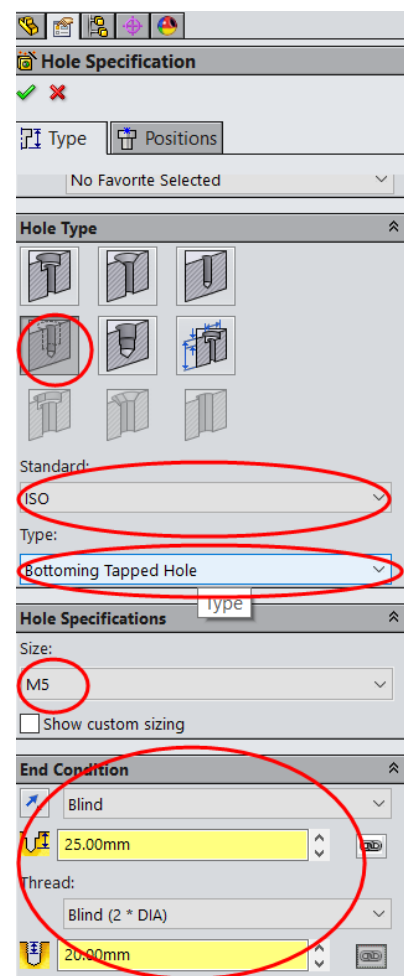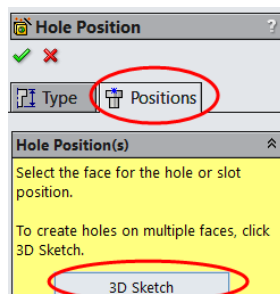
Exit the sketch

In the features tab, select **Hole Wizard**
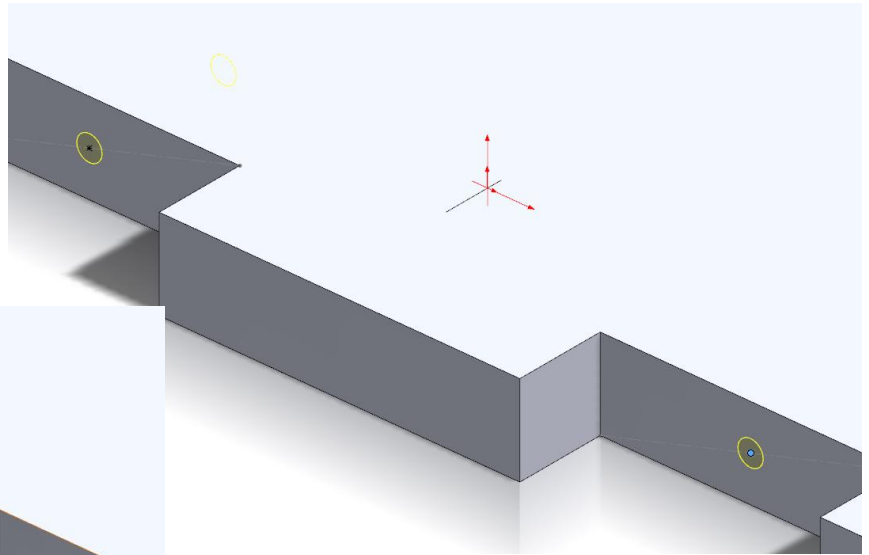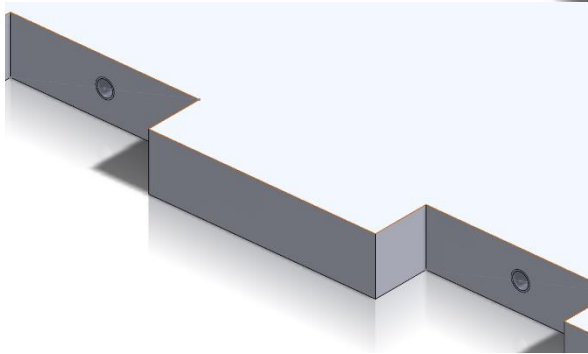
The following specification is added:

- Type: **Straight Tap**
- Standard: **ISO**
- Hole Specification; **M5 x 0.8**
- End Condition: **Blind, depth 25mm, thread 20mm**
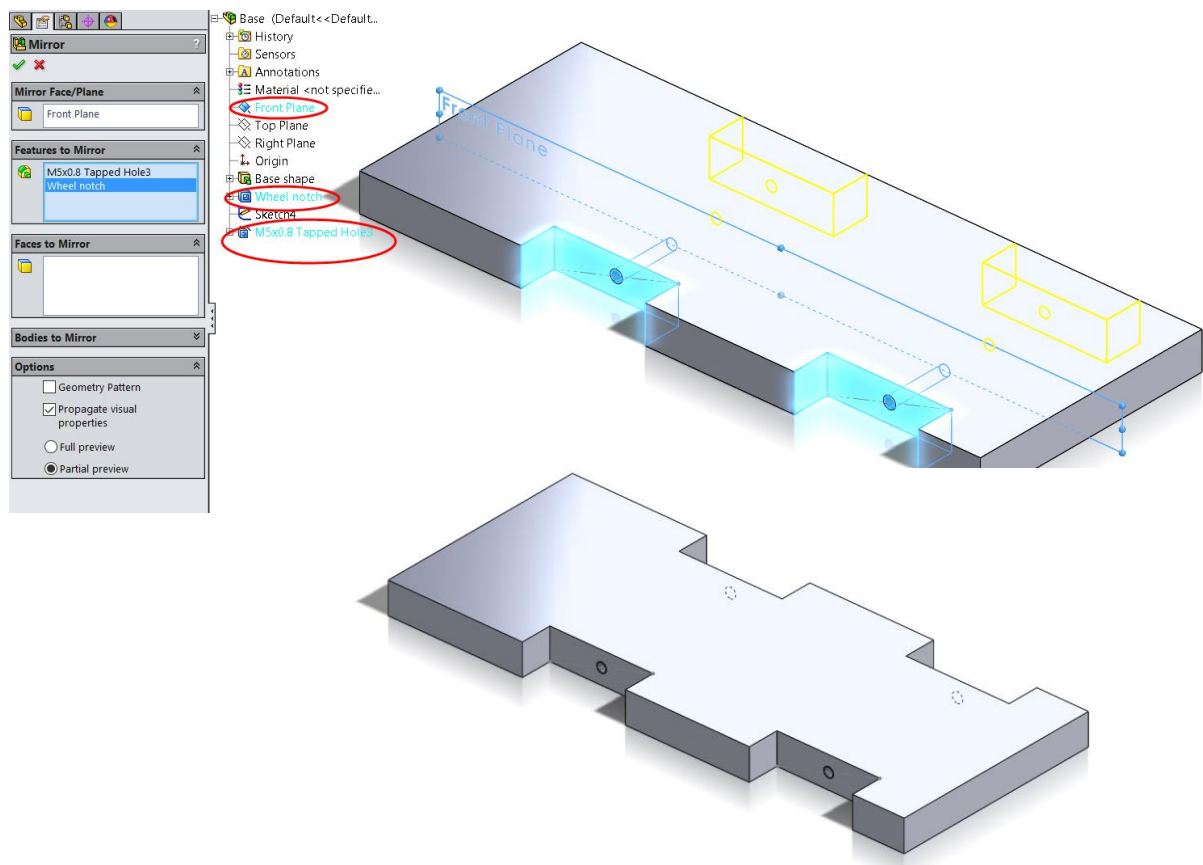
Then select **Positions** and select **3D Sketch**

Select the two points the sketch for the position of the Hole

Click OK ✅ and save


Mirror the Features

Select **Mirror** from the features tab. Open the design tree (press the + symbol beside

⊞ 🗊 Base) and select the **Front Plane** as **Mirror Face** and the **wheel notch** and **Tapped Hole**
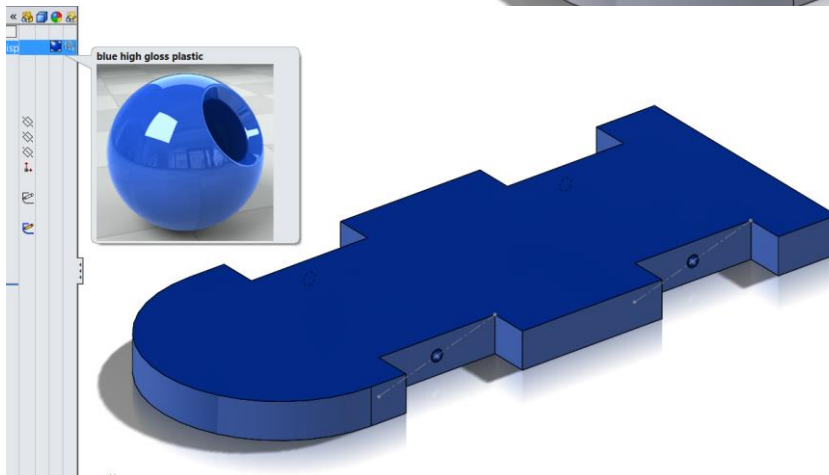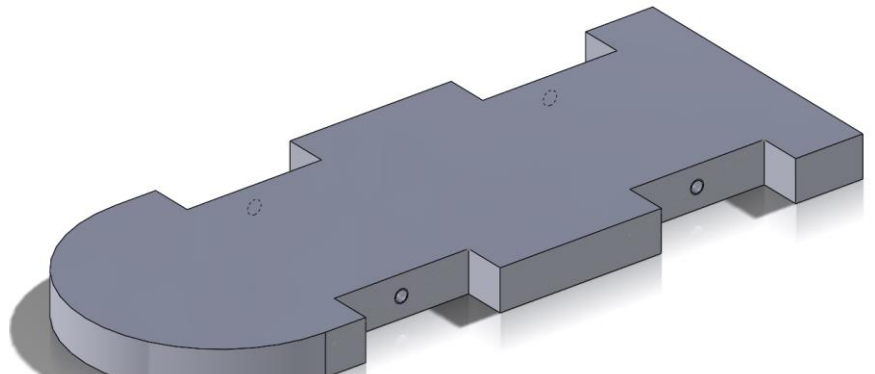
as the **Features to Mirror.** Click OK ✅

Full Round Fillet

Select the Fillet feature ![Fillet]. Select the **Full Round Fillet** option and select a face to enter into each of the boxes below, working left-to-right or vice versa. Click OK ✔
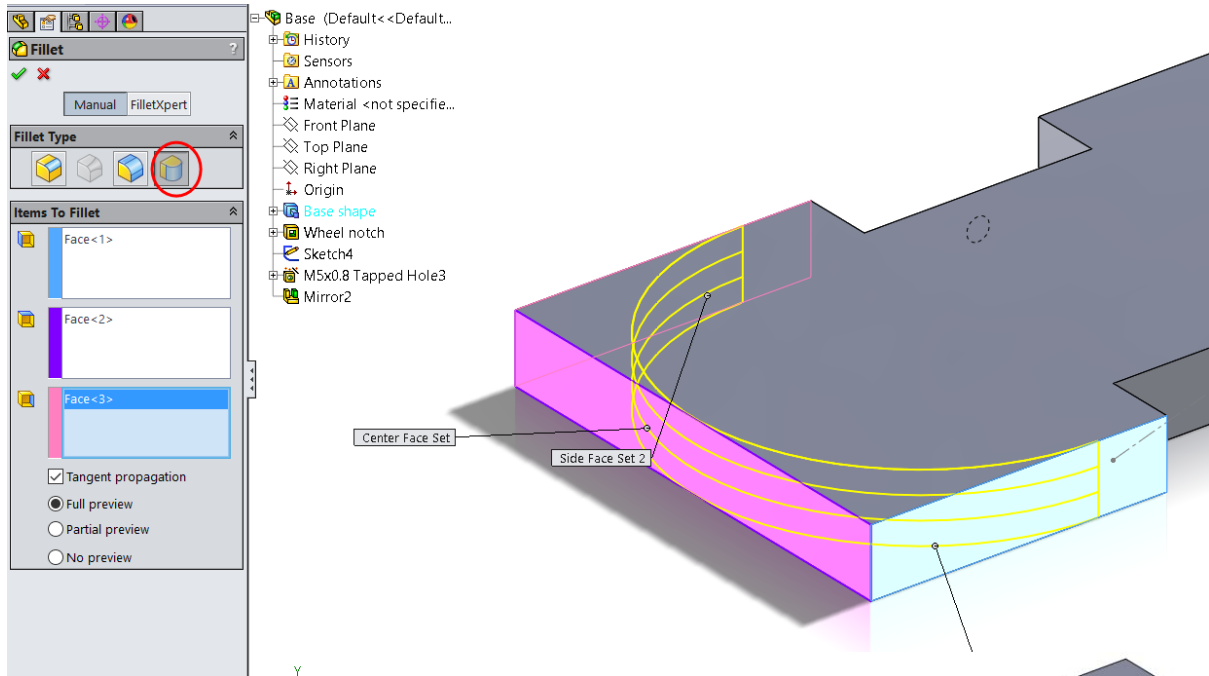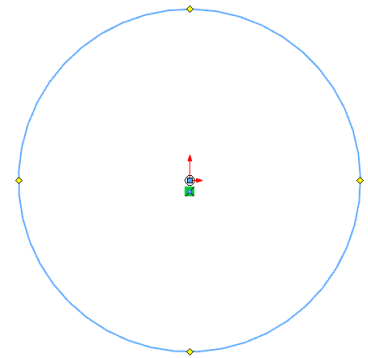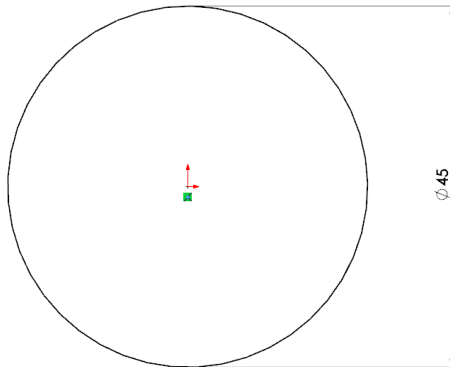

Add a **blue high gloss plastic**

appearance to the part and save.
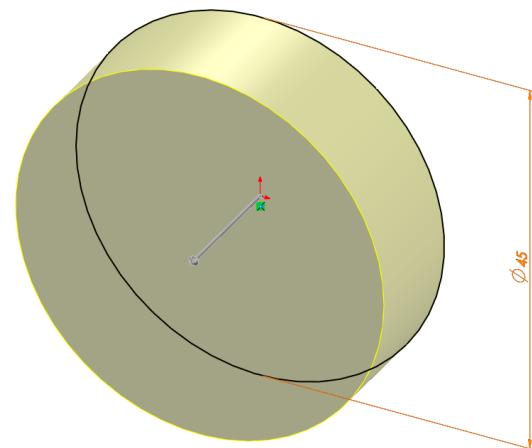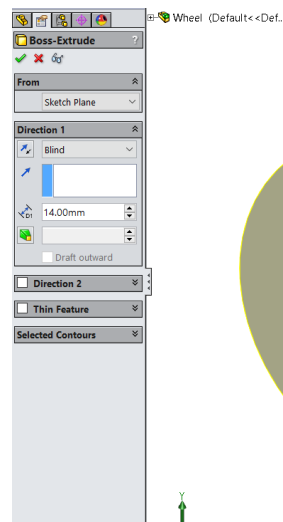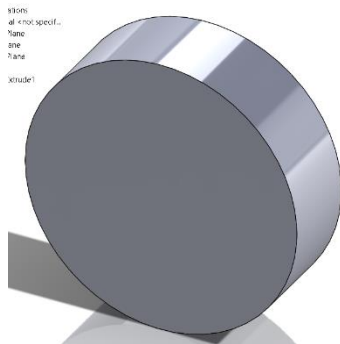

The Base part is now complete

Part Modelling: Wheel

Open a new part and save as **Wheel.** Create a sketch on the **Front Plane.** Using the circle command ⊙ ▾ draw a circle with its centre on the origin. Smart Dimension the circle as **45mm**.
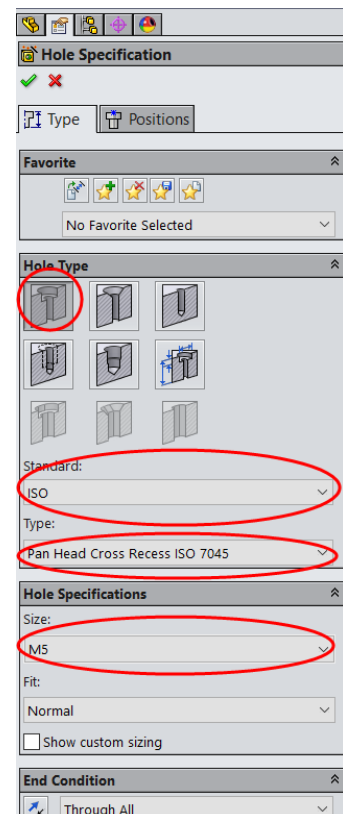
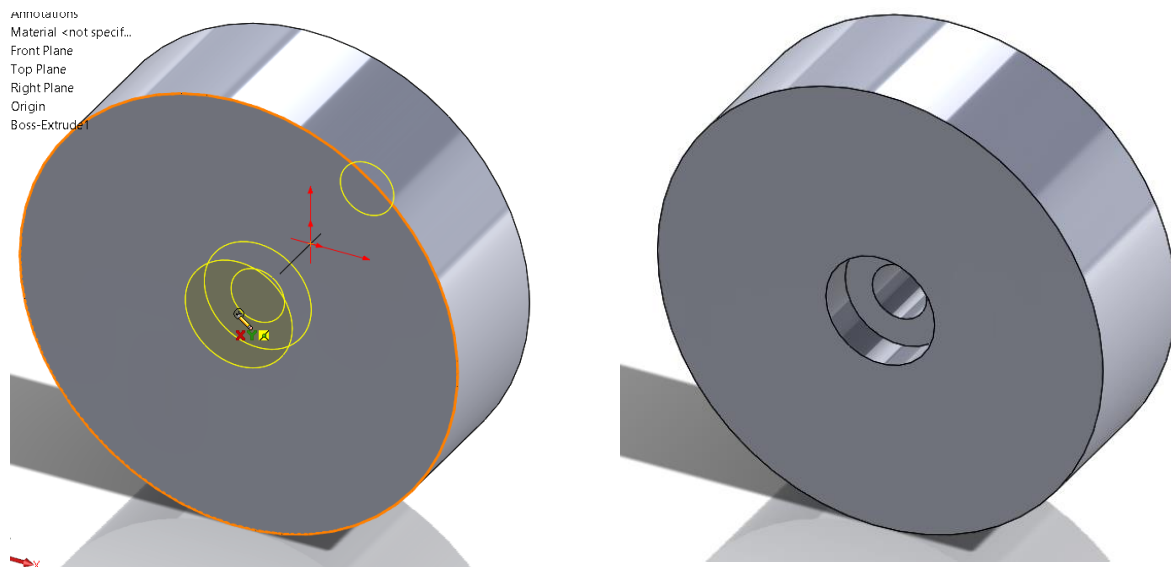Extrude 🗐 Extruded Boss/Base the circle **14mm**

Click OK ✓

Select the Hole wizard feature and edit the specification

- Hole type: **Counterbore**

- Standard: **ISO**

- Type: **Pan Head Cross Recess**

- Size: **M5**
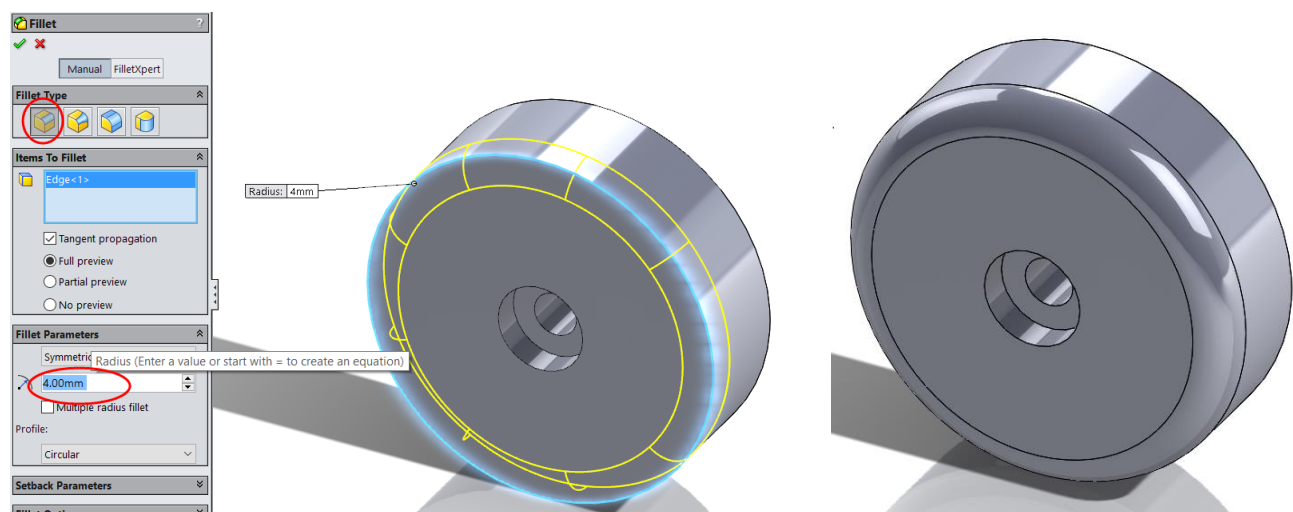
- End Condition: **Through all**

Select **Positions** and **3D Sketch** and hover over the circle face until centre-point appears.

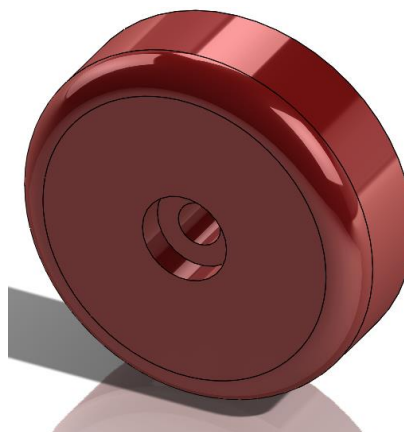Click on the centre to add the hole. Click OK ✅

Select the Fillet command ![Fillet] and add a **4mm Constant Size Fillet** to the front edge and

click OK ✅. Save your work

Add a **Red high gloss plastic** appearance to
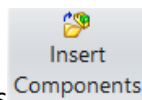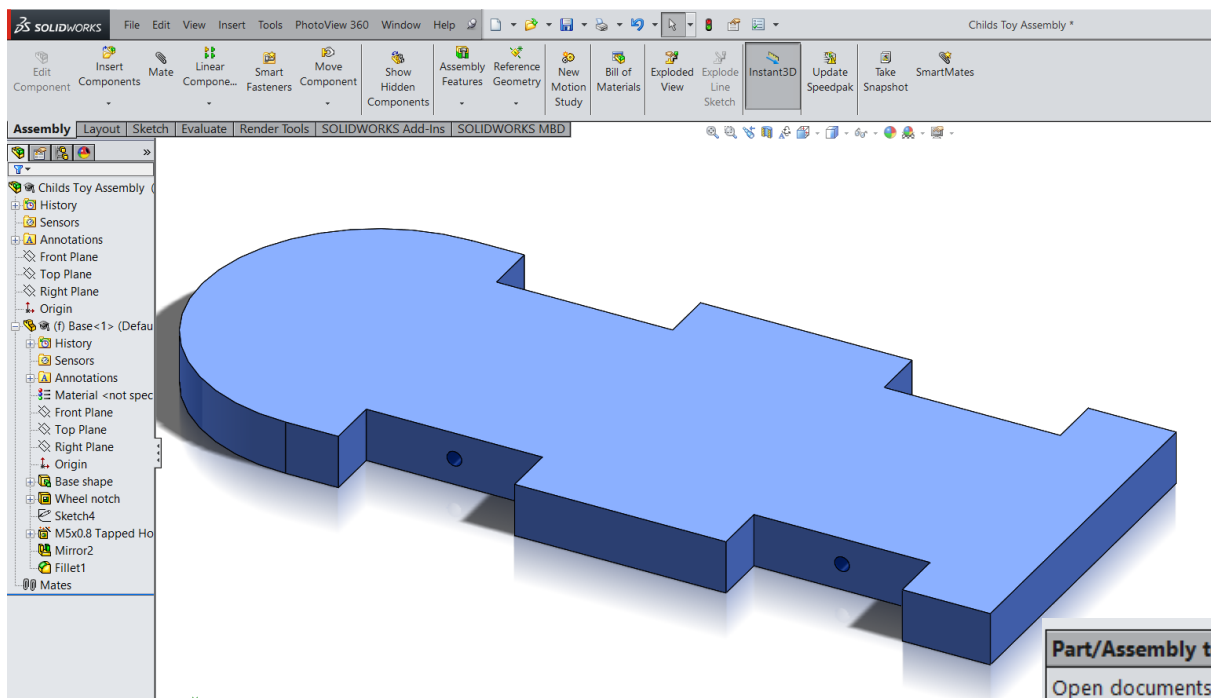
the wheel
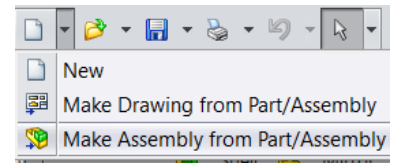
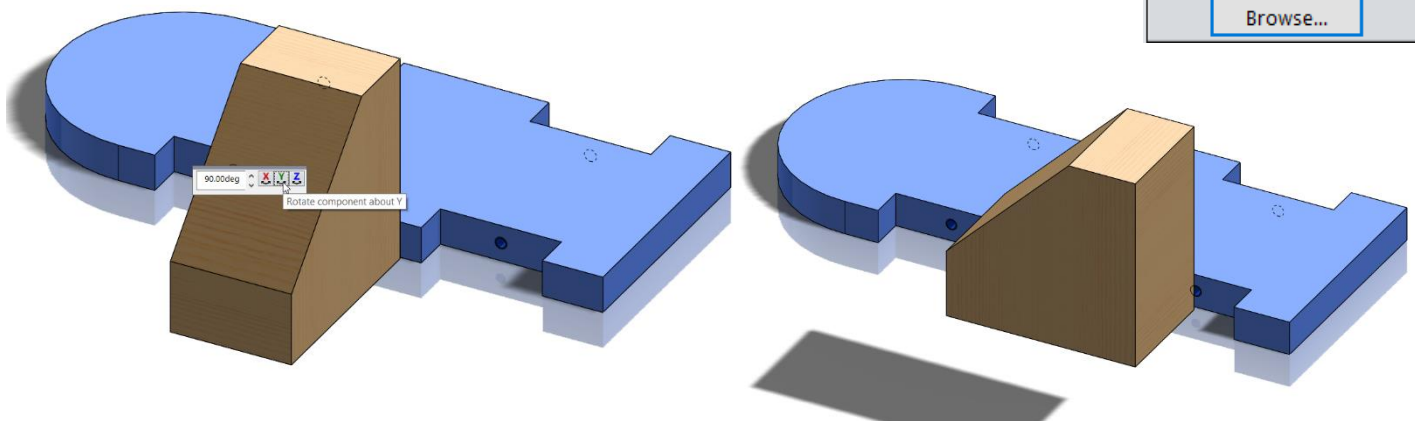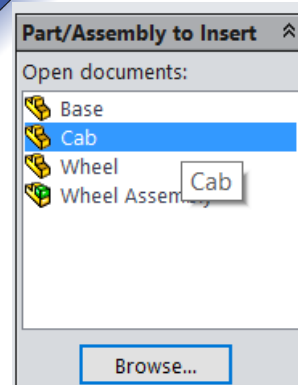The Wheel part is now complete.

Assembly: Base and Cab

Create a new assembly or within the **Base** part select Make

Assembly from Part/Assembly

Insert the base part and select OK . Save as **Childs Toy Assembly**


Select Insert Components and (**Browse** if not shown) click the **Cab**

part. Use the rotate on insert tool to rotate about Y axis into the correct

orientation. Click OK

Rotate the view (hold down middle mouse button) and select the bottom of the cab, on the pop-up toolbar select **Mate**. Select the top surface of the Base and add the coincident mate
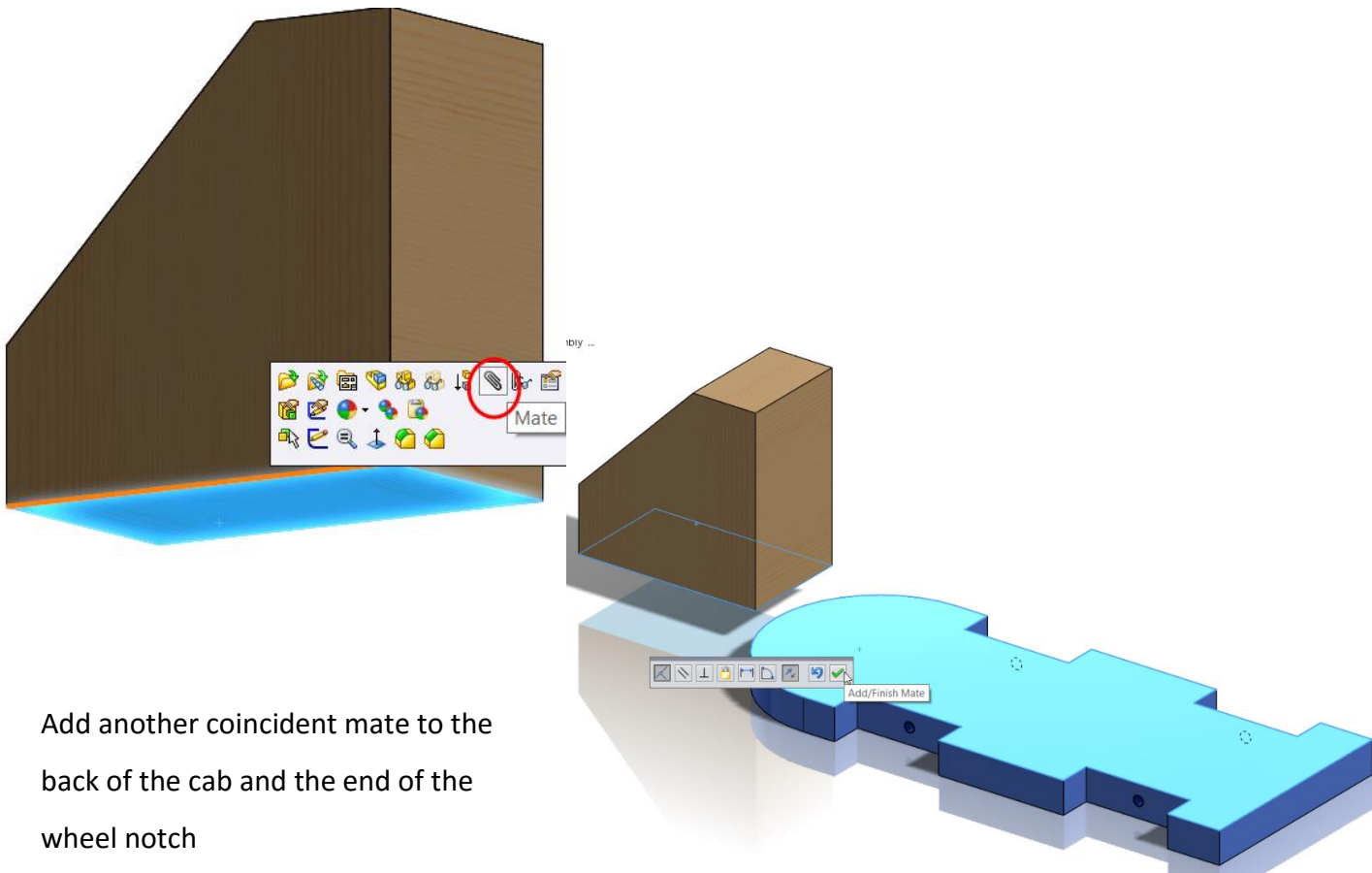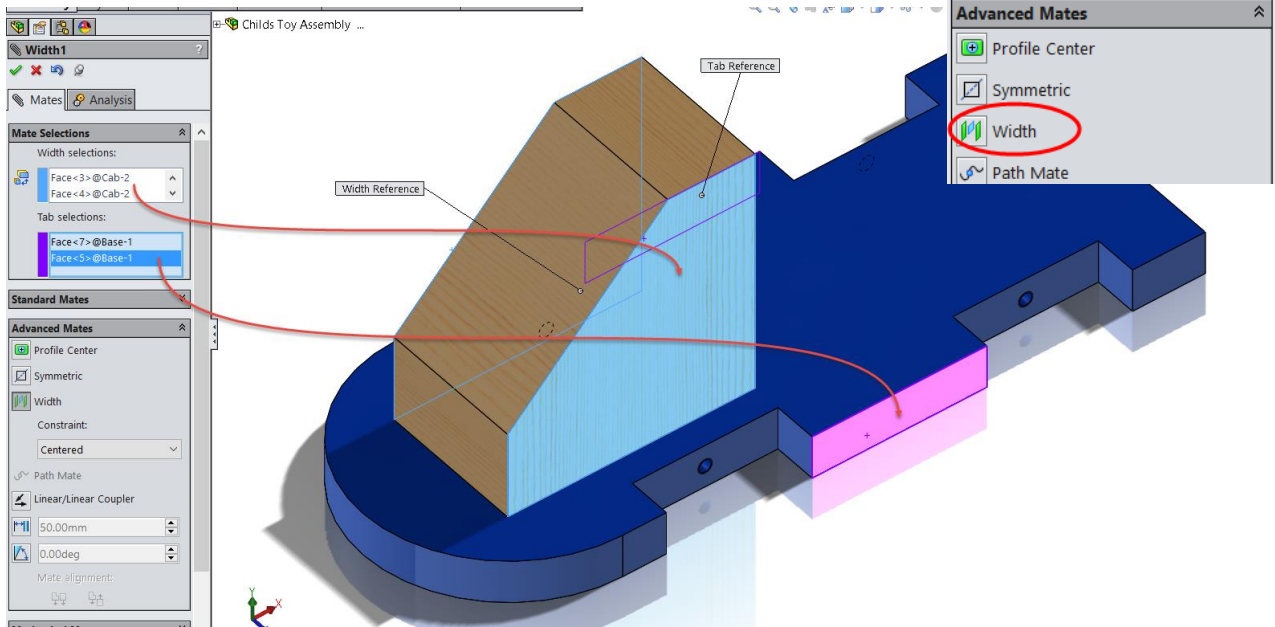

Add another coincident mate to the back of the cab and the end of the wheel notch
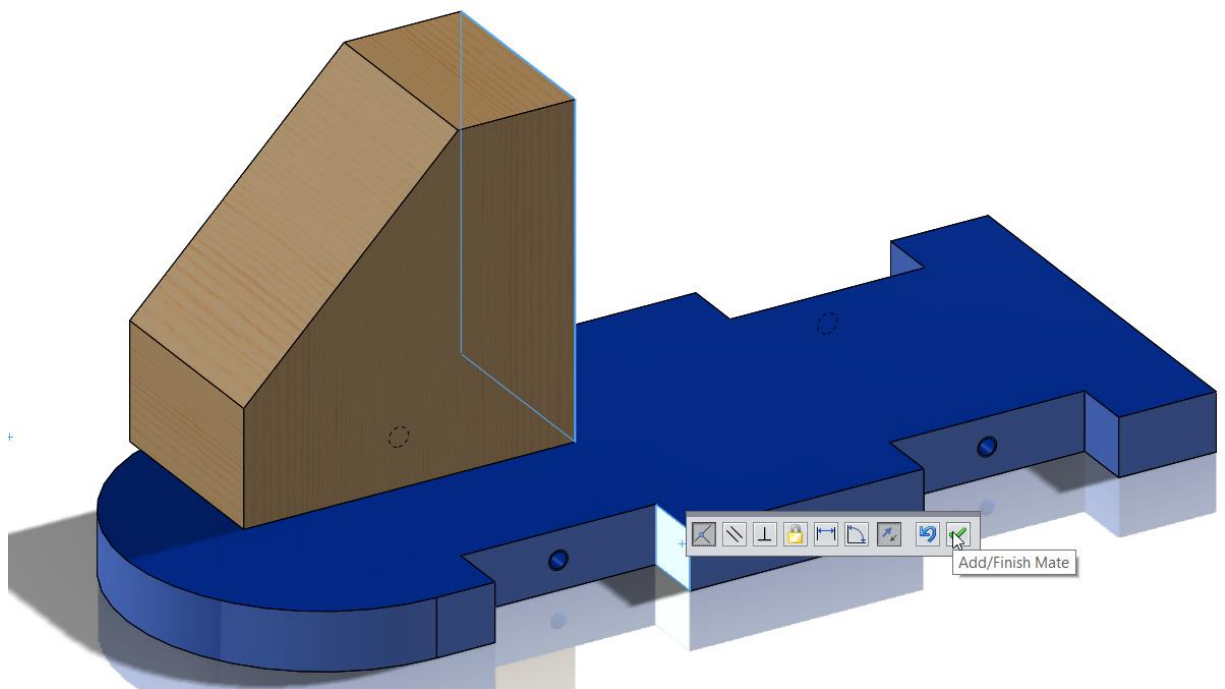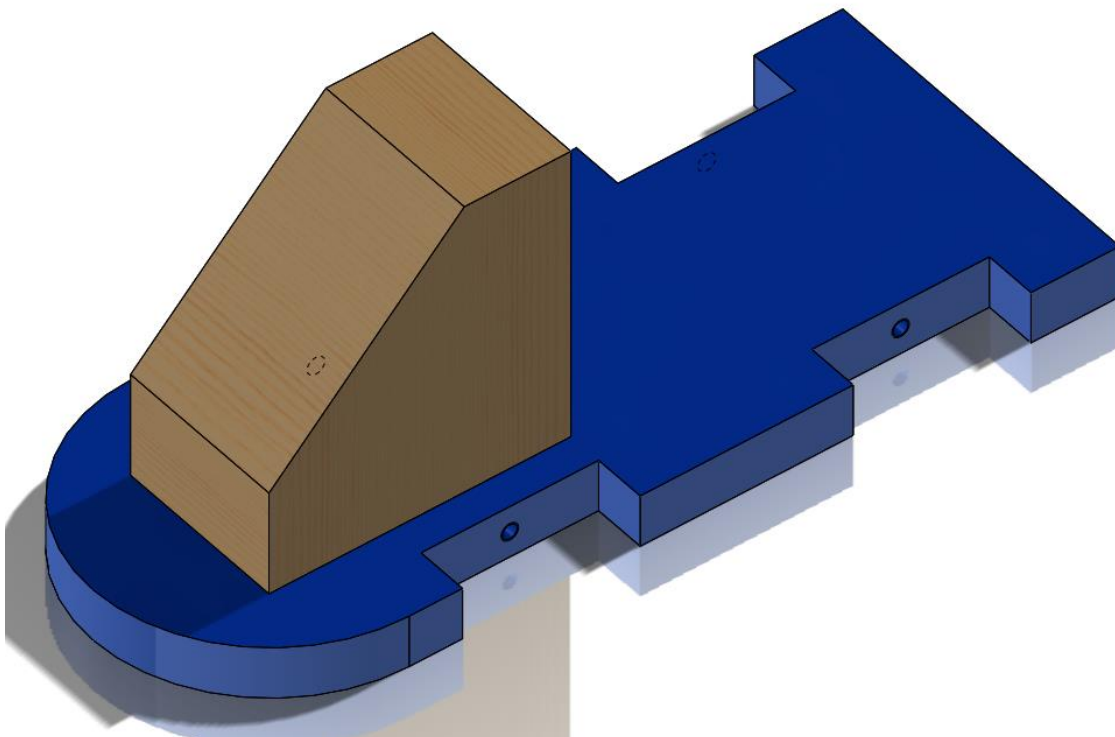
In the **Advanced Mates** tab select **Width** mate. In the Width made select the two sides of the cab for the width and the two sides of the base for the tab.
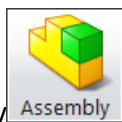
Click OK ✅ when complete

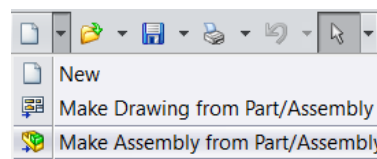Save your work. The first part of the assembly is complete
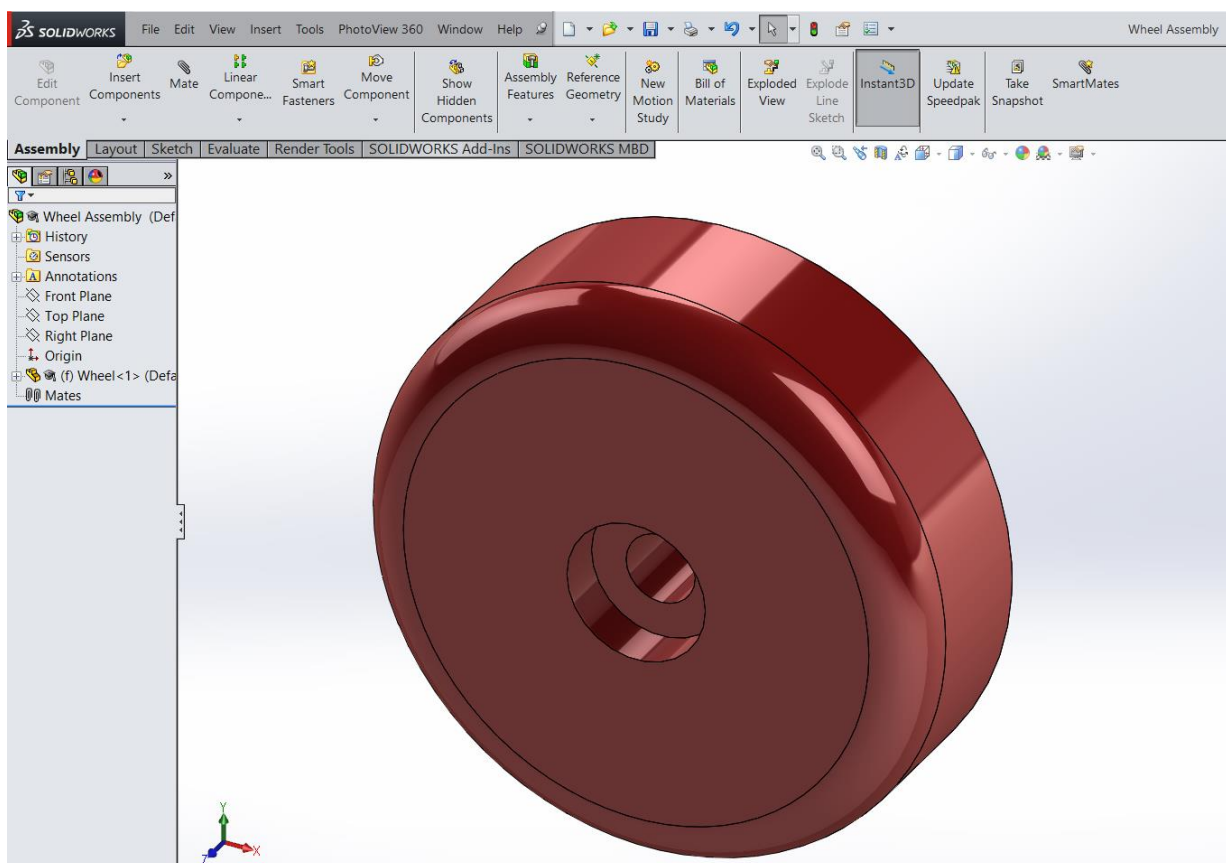
Assembly: Wheel Assembly

using the Toolbox

Open a new assembly [Assembly] or within the wheel part select **Make Assembly from Part/Assembly**. This assembly will be used as a sub-assembly for the Toy.

Select OK ✔ in the new assembly file. Save as **Wheel Assembly**
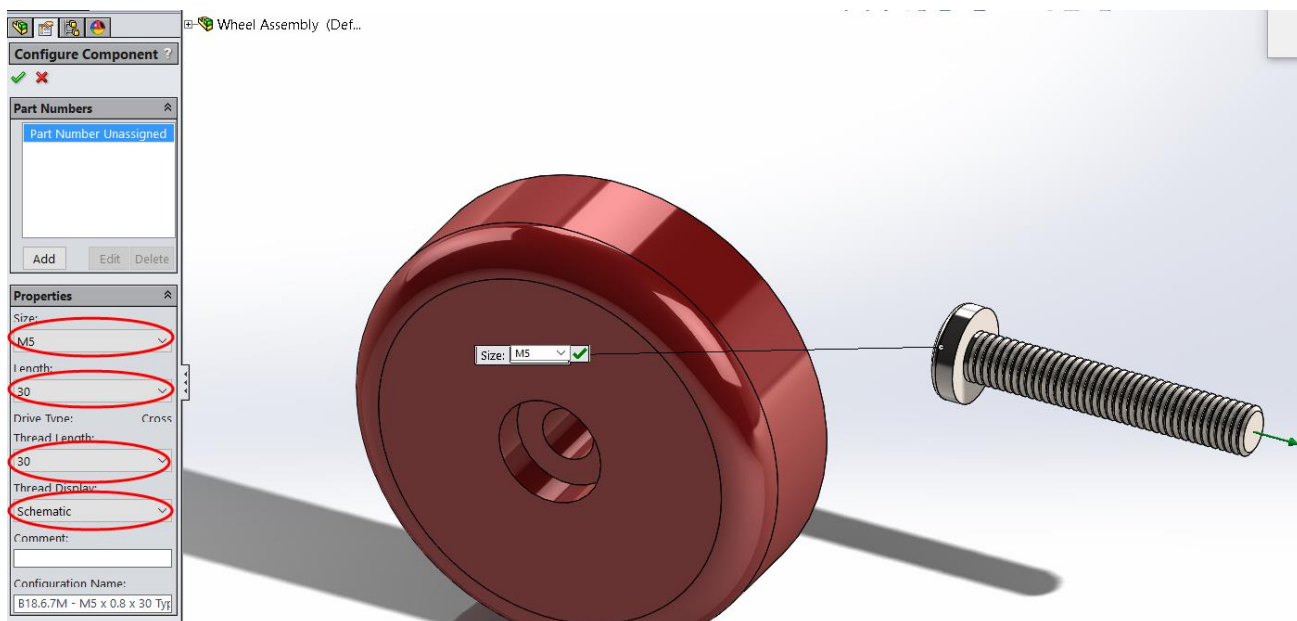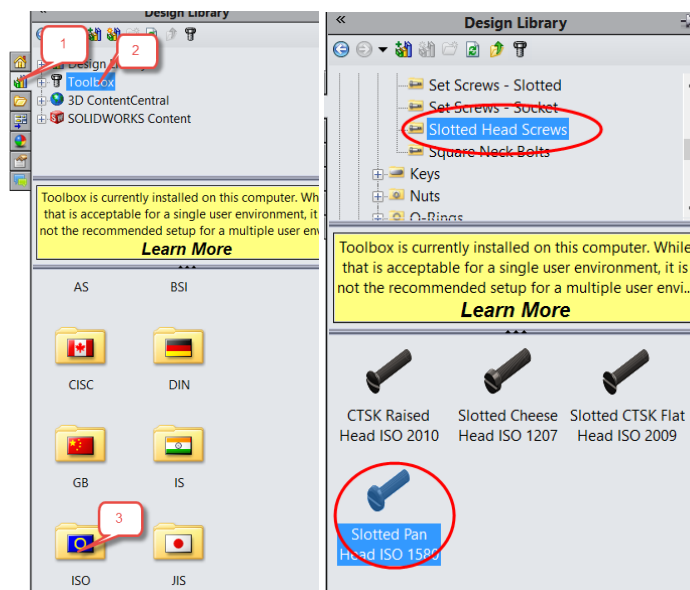
Using the Toolbox.

Select the Design Library tab and then the
Toolbox icon. (If it is not added select **Add
Toolbox). Browse for ISO.**

Then browse for **Bolts and Screws**,

Slotted Head Screws and select **Slotted
Pan Head**

Drag and drop screw onto screen

Configure the component to the following
specification

- Size: M5

- Length: 30mm

- Thread Display: Schematic

Click OK, then click X to accept only 1 screw

Mating the Wheel Parts

Select the mate command ⌗ Mate and select the cylindrical surface of the screw head and the hole to add as a concentric mate (automatic). You may need to flip direction ⬈. Click OK to accept mate ✓. Also select **Lock Rotation**
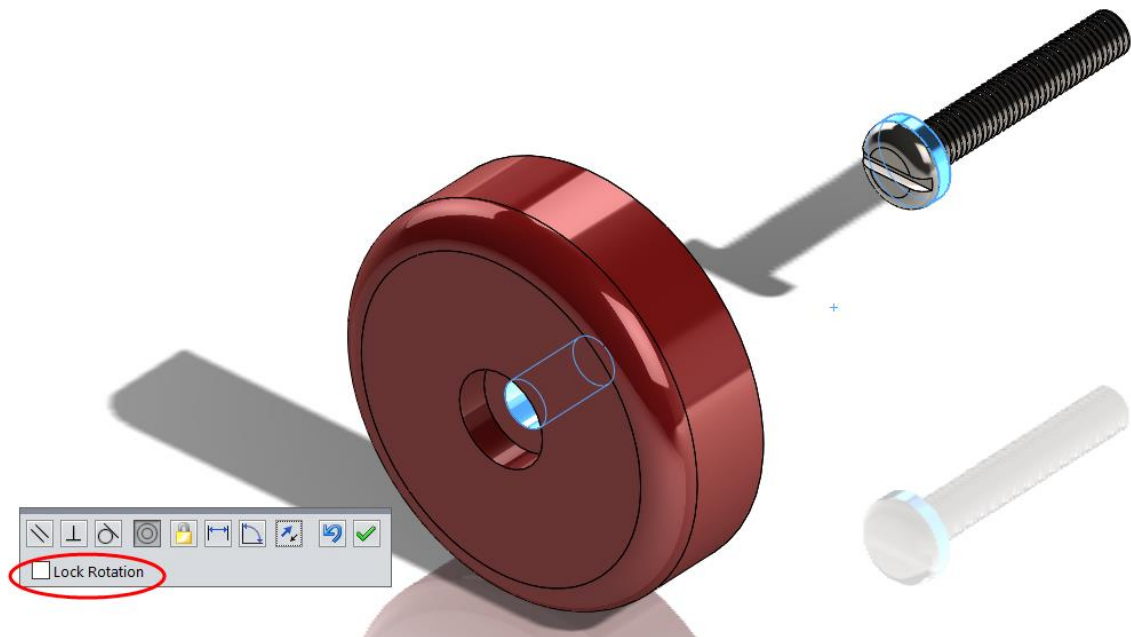
Add a second coincident mate, selecting the back of the screw head and the inner surface of the counterbore. Accept mate and Select OK. Save your work.
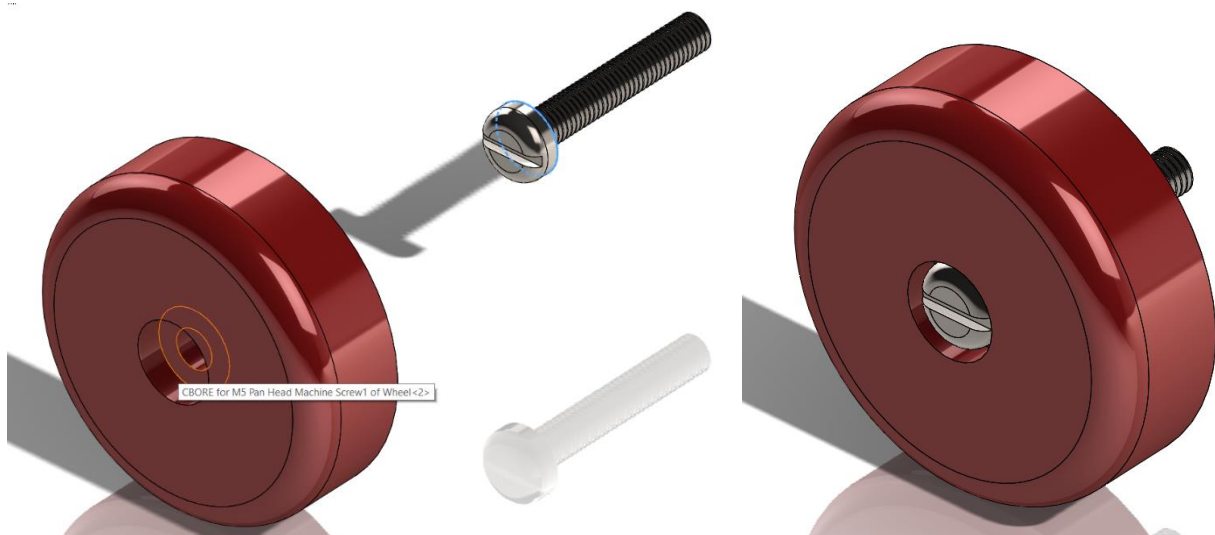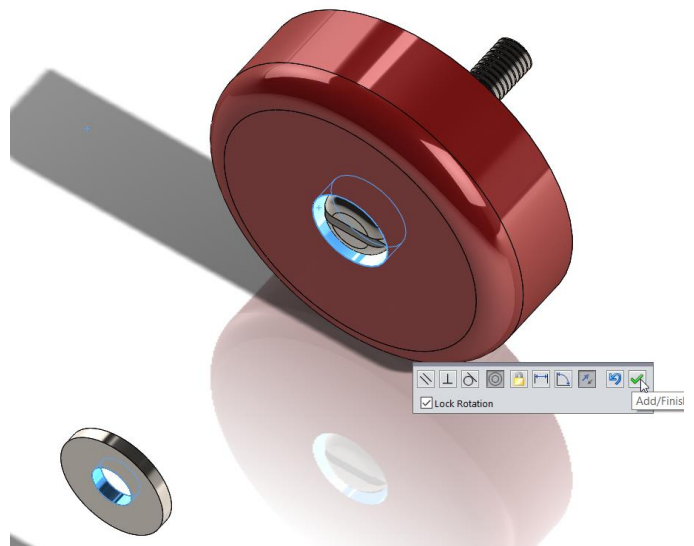
Go to the toolbox and browse for **Plain Washer, Regular Flat Washer..**Drag and drop in a washer. Configure to a **M5** washer. Accept and press X.
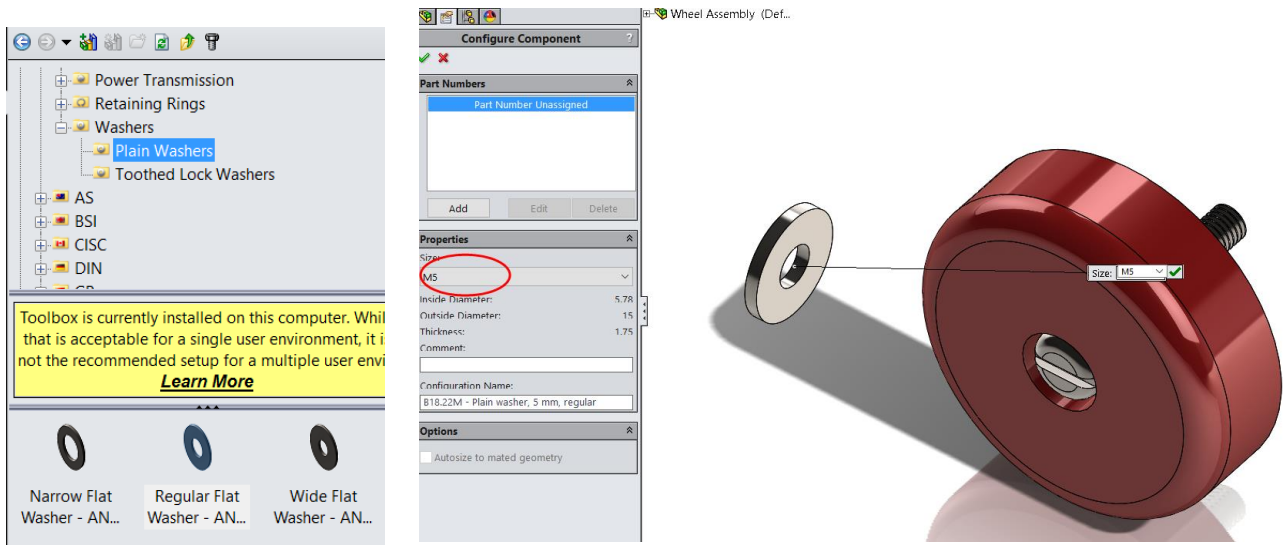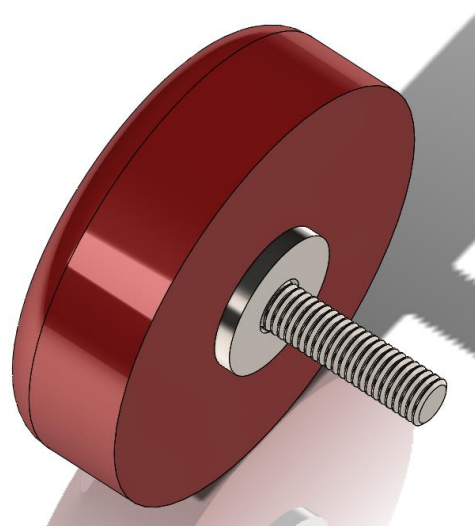

Mate the cylindrical washer surface to the cylindrical surface of the wheel. Select **Lock Rotation** and add this mate. Mate the flat surface of the washer to the back of the wheel.

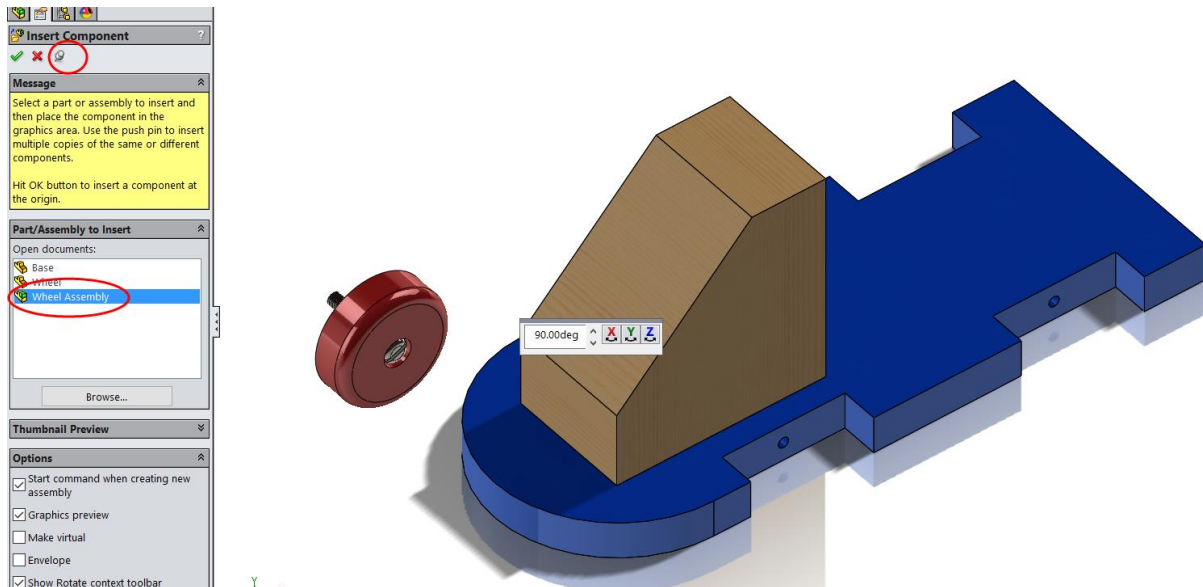Click OK to exit the mate command

Save your work
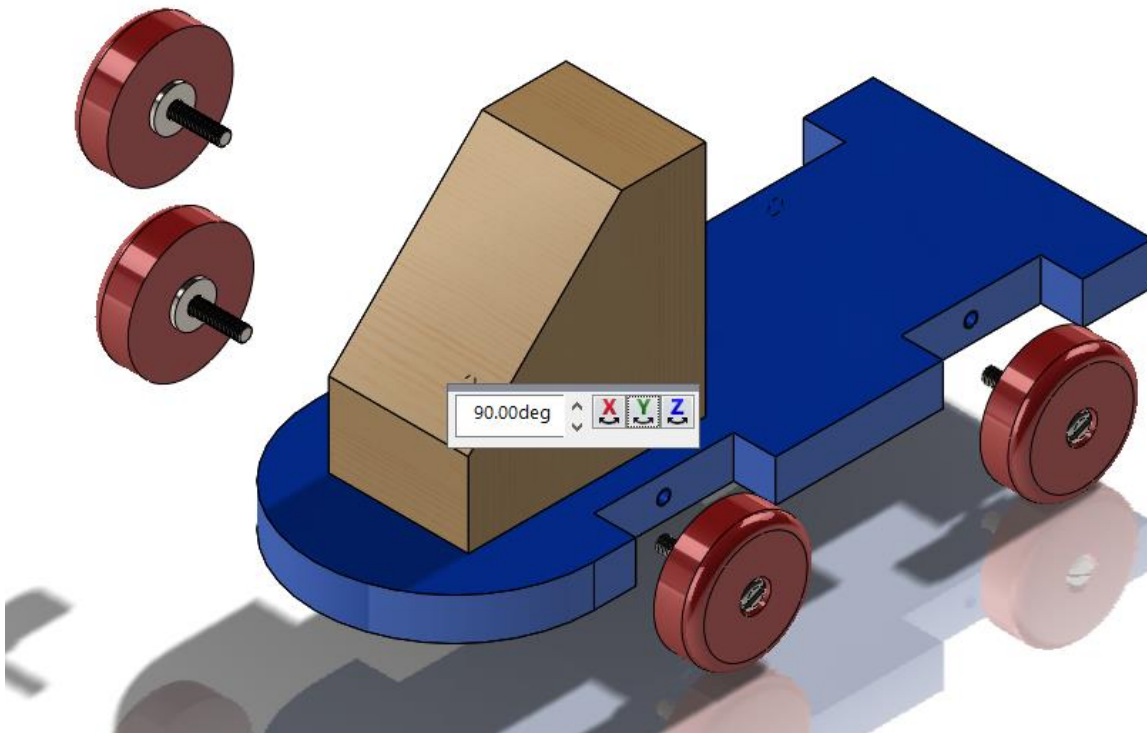
The **Wheel Assembly** is now complete

Completing the Assembly

Insert Wheel Assembly

Open the Child's Toy Assembly and select Insert Components. Select the Wheel Assembly to insert. Press the **Keep Visible** pin down to allow you to insert the assembly multiple times.
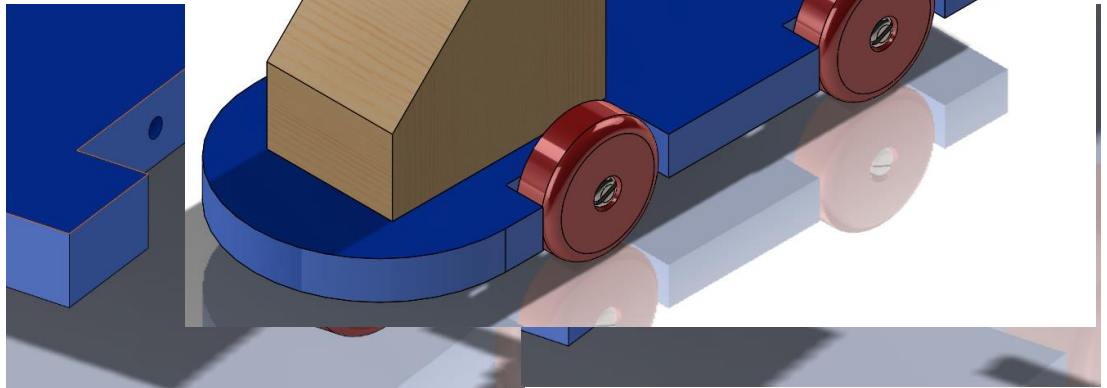

Use the rotate on insert toll to rotate 2 wheel assemblies about the Y axis

Click OK ✅ to finish the insert command

Mating the Wheels to the Base

To add mates between the screw and the base, hide the wheel; press the **TAB** key down and hover cursor over wheel.
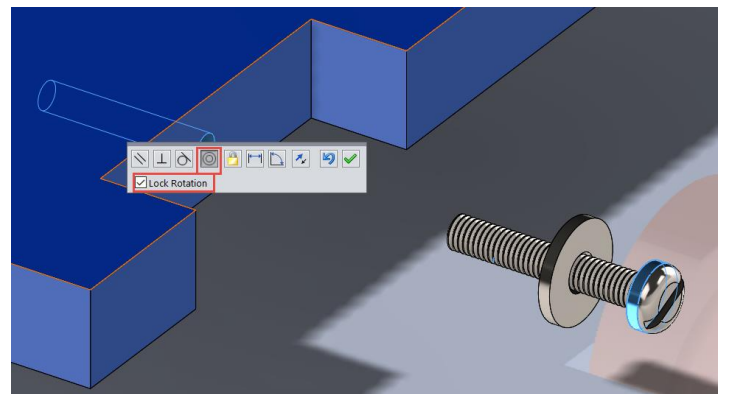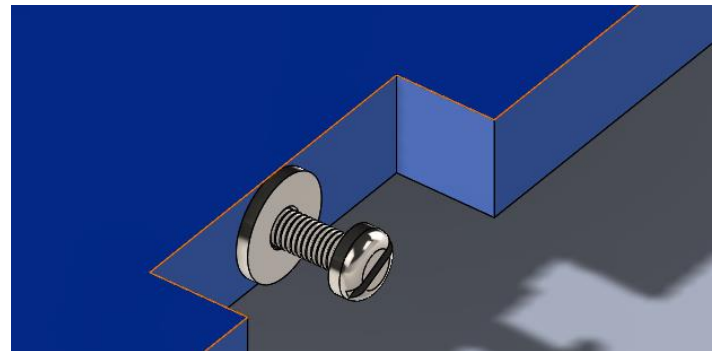
Mates to add:

- Concentric: cylindrical surface of screw and Tapped hole
- Lock rotation
- Coincident Back of washer with notch surface

Show the wheel again by holding down **SHIFT + TAB** on keyboard and hover over where wheel is.
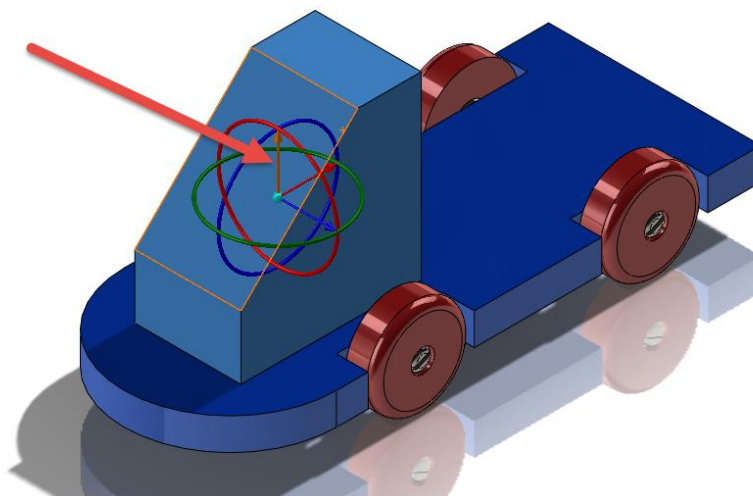
Repeat this for each wheel assembly

Save your work. The Assembly is now complete
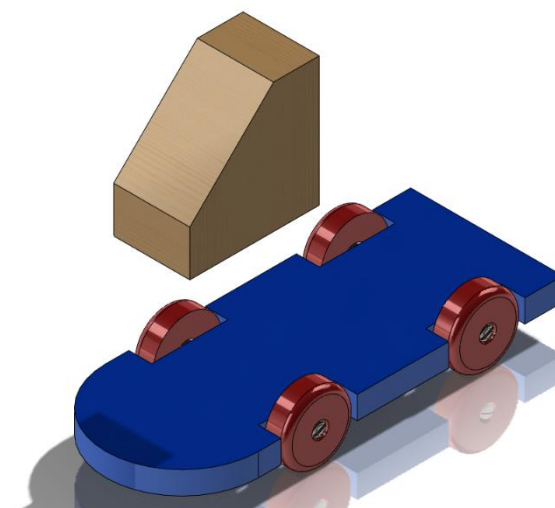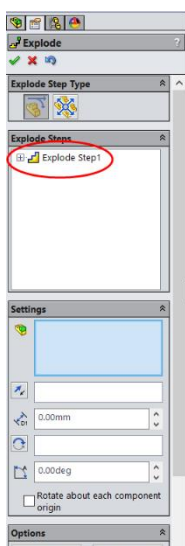
Creating an Exploded View
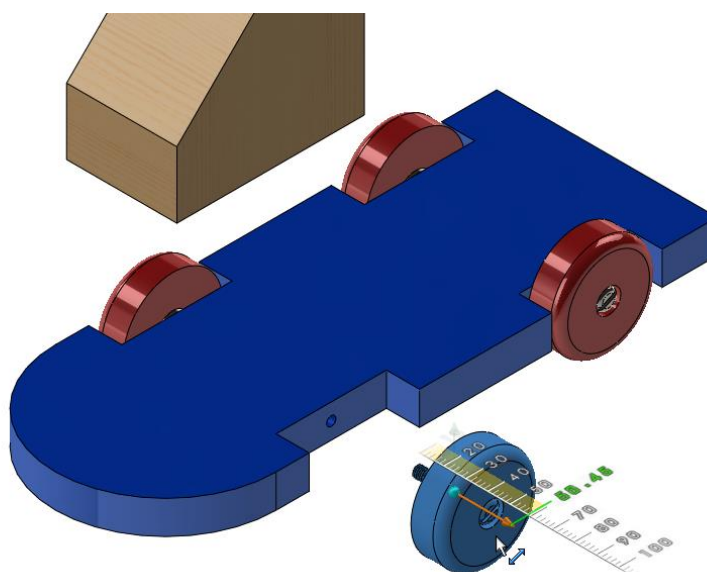
Select **Exploded View**

. Click on Cab and
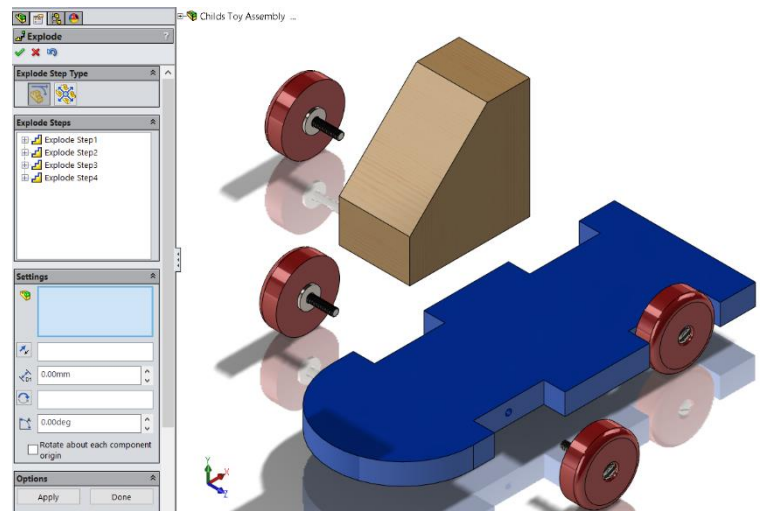select Y (vertical) arrow to
move the Cab.

Pull the Cab upwards. When
in place click anywhere away
from assembly. Note this is
now Explode Step 1

Select the Wheel to move next.
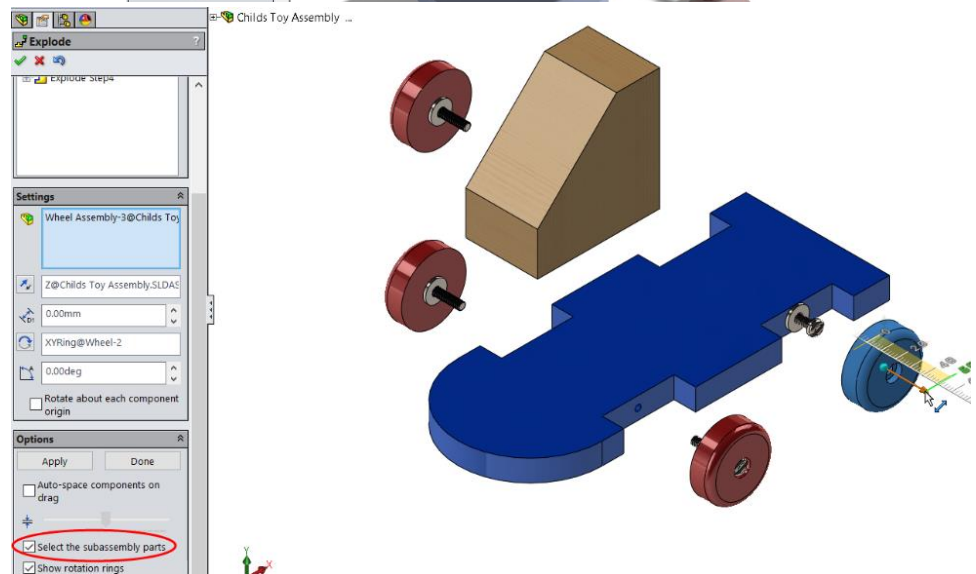Move along the X axis. Note
how the whole wheel assembly
moves
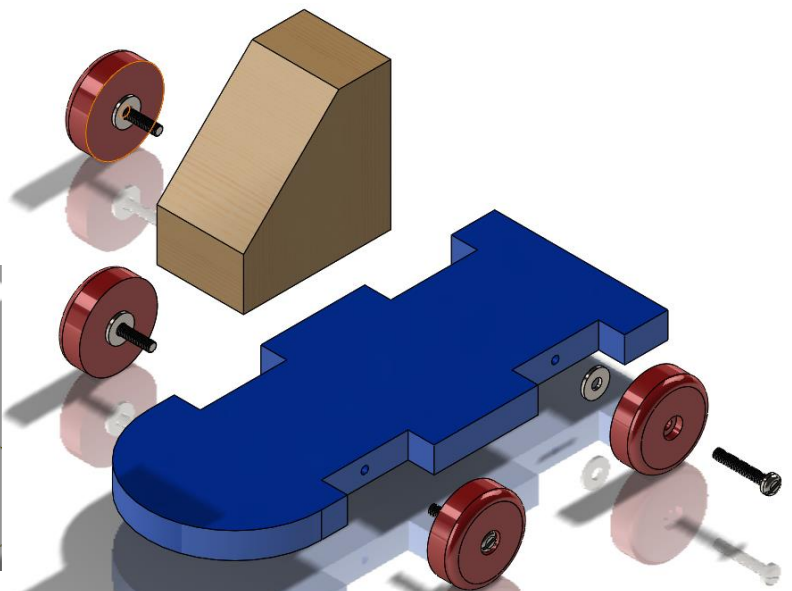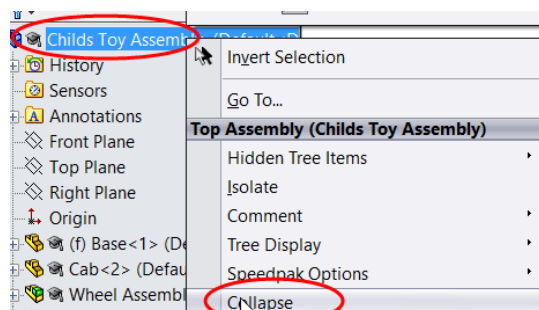
Repeat for the two far wheel assemblies

To explode the wheel assembly for the last wheel, tick **Select the subassembly parts.**

This will allow you to explode the screw, washer and wheel individually
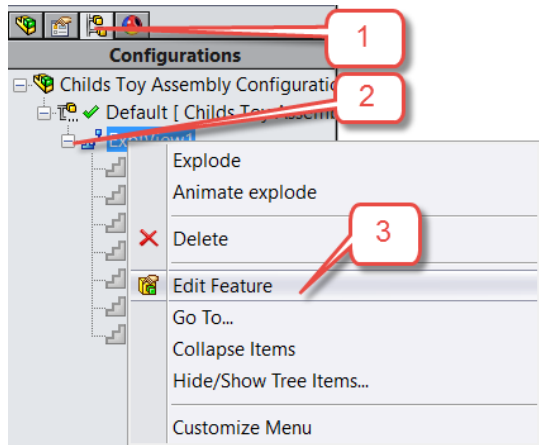
Click OK ✓

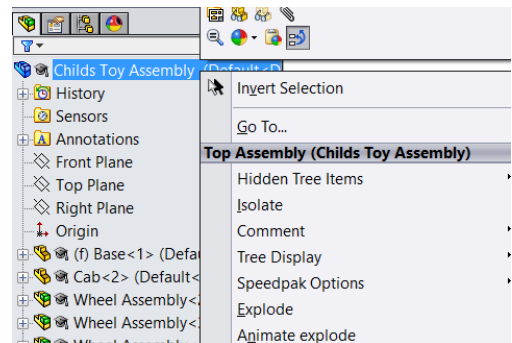To return to the "assembled" view, right click on the top of the design tree and select **Collapse**

To explode again right click on top of design tree and select **Explode**

To edit an exploded view, click on the **ConfigurationManager** tab and select the **ExplView1**, then select **Edit Feature**

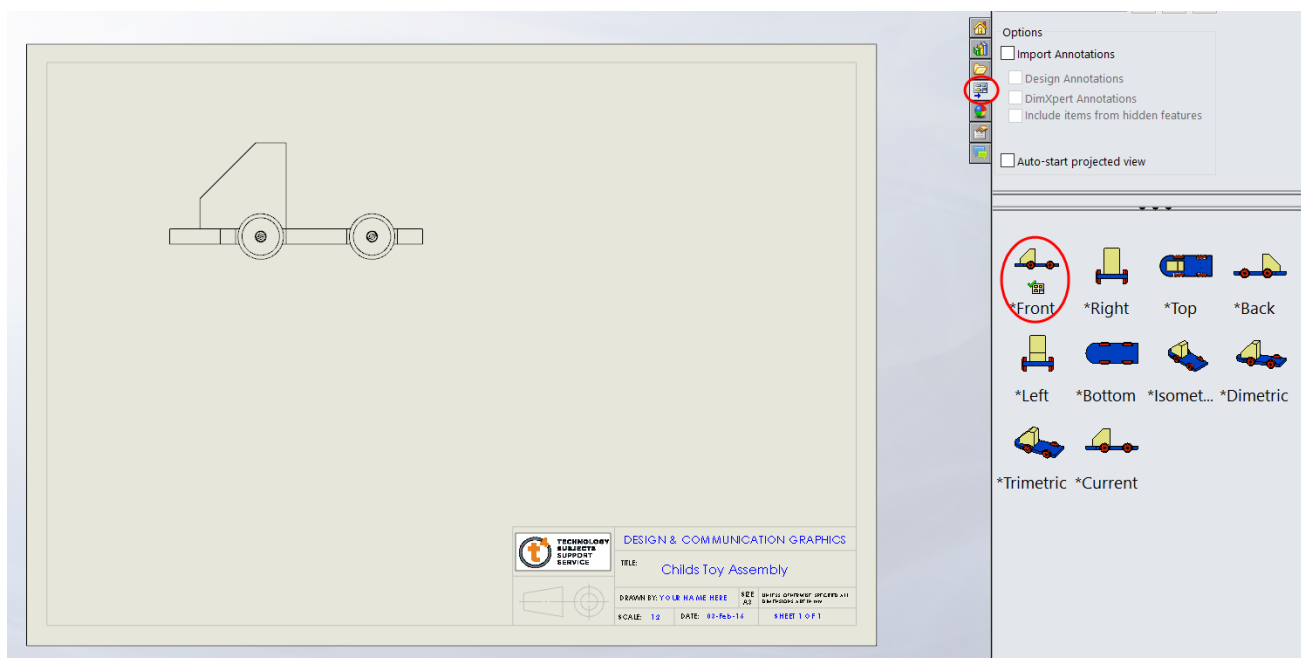Childs Toy Assembly

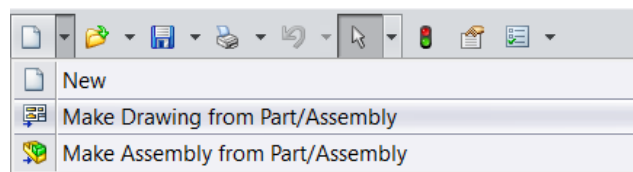# Creating Drawings and Worksheets

Adding Views

Create a drawing from the **Child's Toys Assembly**

Select an A3L sheet template (e.g. **DCG A3L**)

In the **View Palette**, drag and drop the elevation by selecting **Front** view. Drag and drop the

Top view for plan and Left view for end elevation. Notice how the views will auto project from elevation when dragged nearby.

Change to a "colour" view by clicking on the view and selecting **Shaded With Edges**
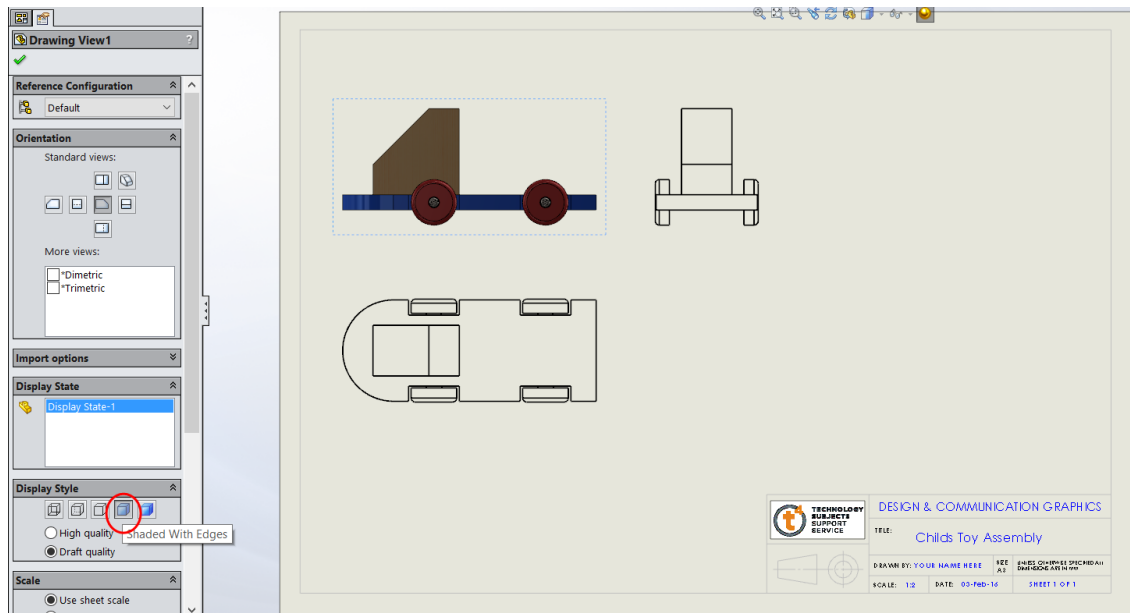
Create the isometric View
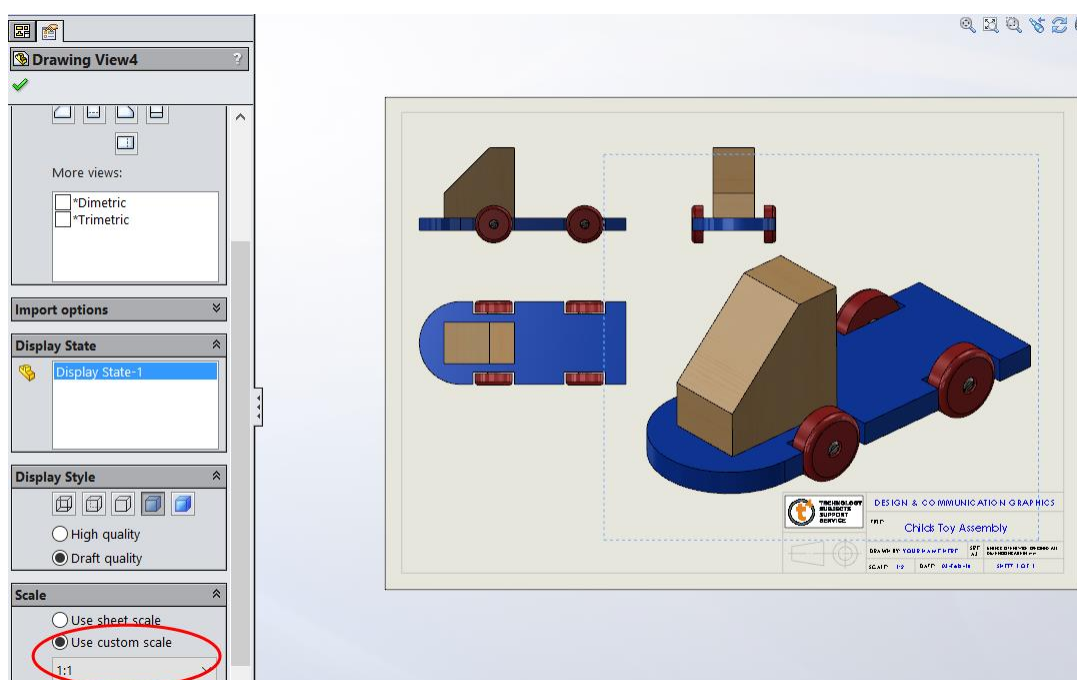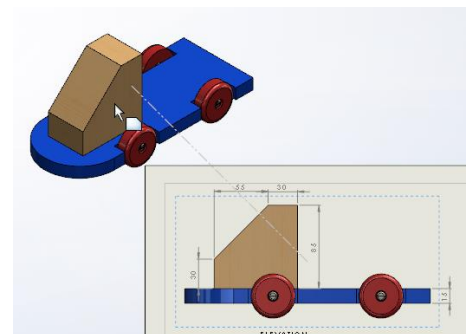
Click on the elevation and select Projected view from View
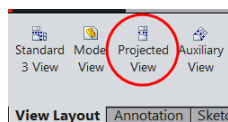
Layout tab.

Project out to the top lef. Hold the **CTRL** button down to move the isometric into the correct position.
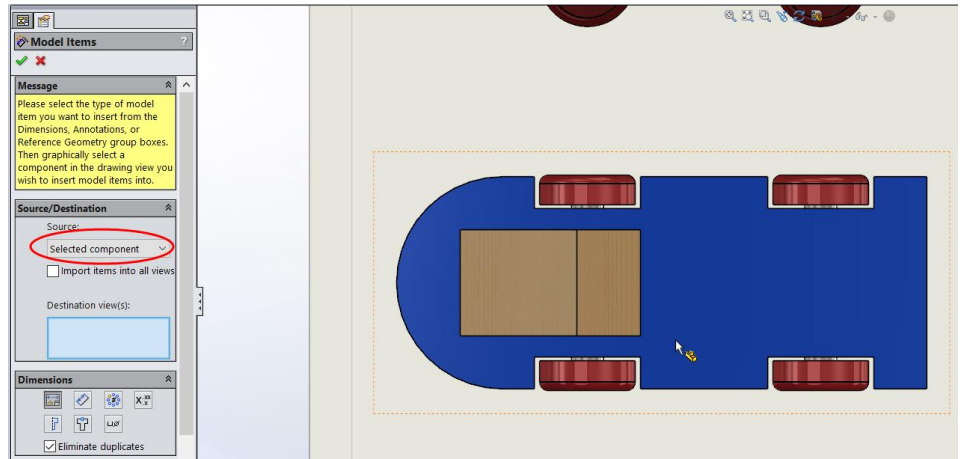
To change scale from1:2, click on view and change. **Use custom scale** to **1:1**
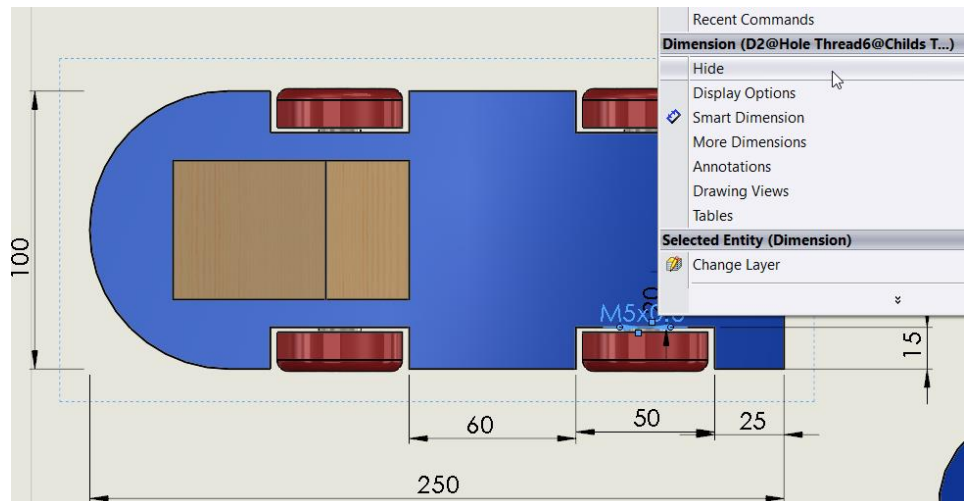
Dimension Drawing

Select **Model Items** from **Annotation** tab. Select **Selected Component** as the Source and click on the Base in Plan. The Import items into all view is un-ticked..
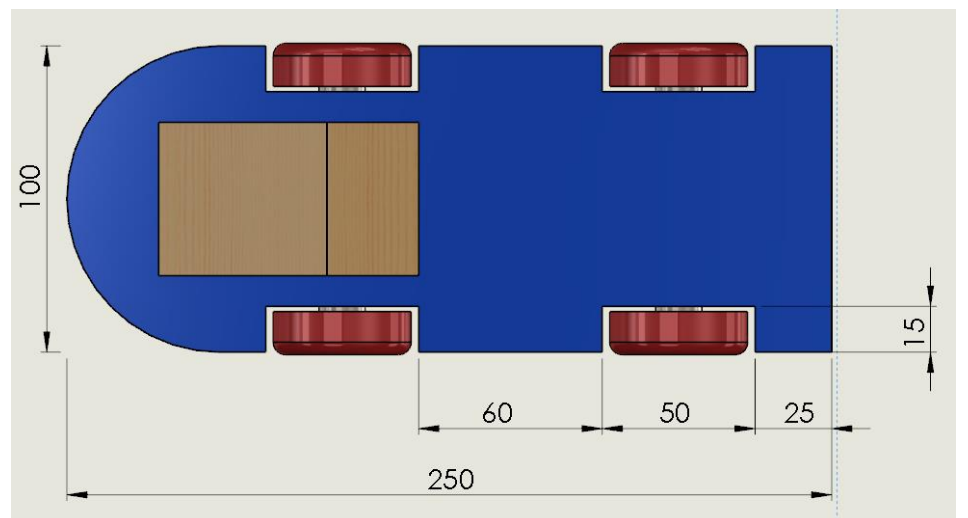

Note: These dimensions are *Driving dimensions* imported from the parts
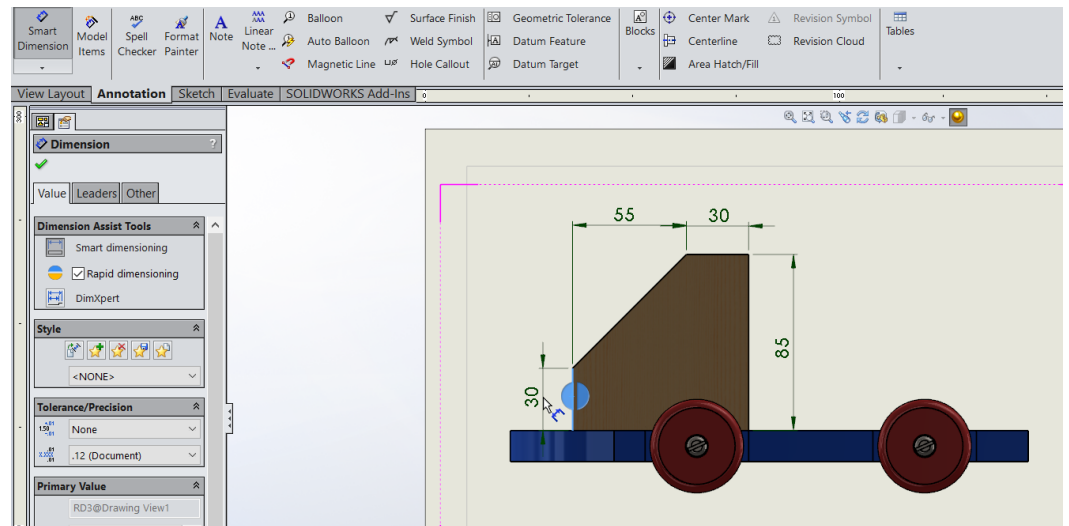

Right click on any dimension you don't want and **Hide**

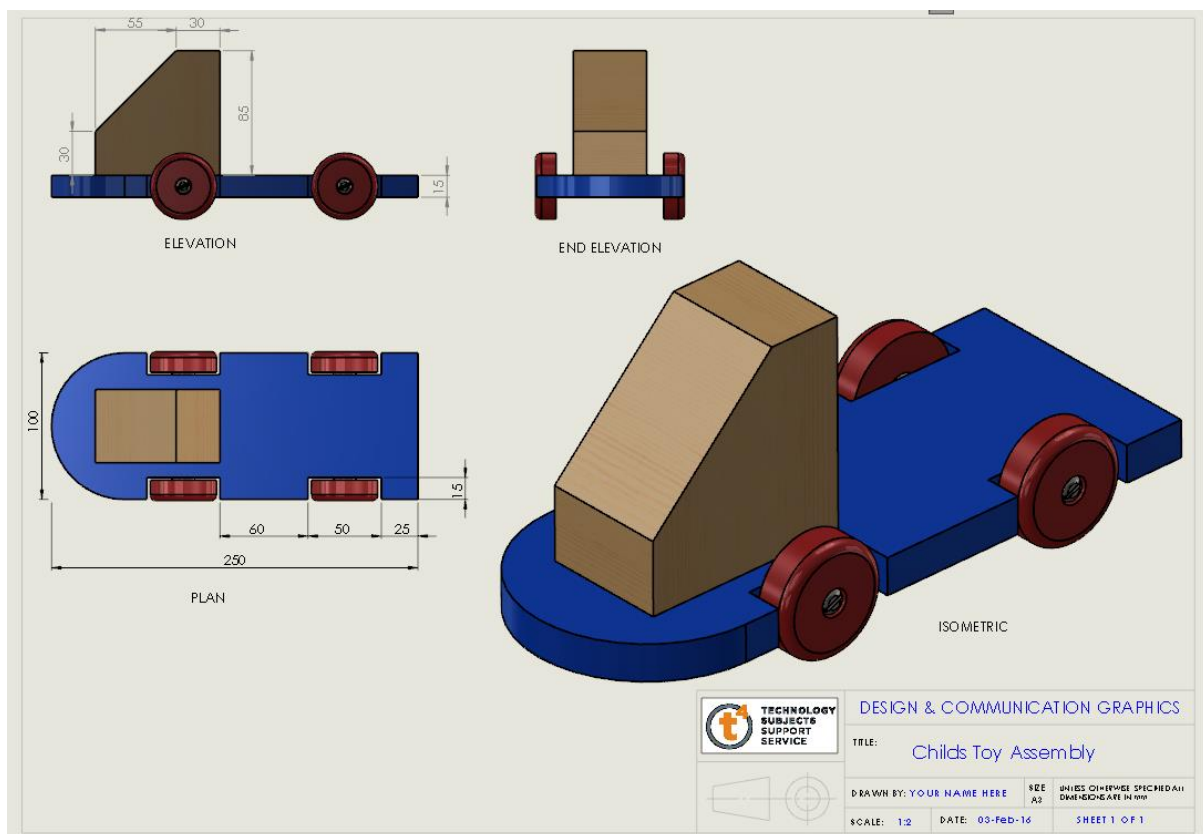Adjust the dimension to appear as neat as possible on the Plan

You can also add driven dimensions by clicking on Smart dimension and add dimensions "manually"

Note: these dimensions are *Driven dimensions*; non-imported from parts

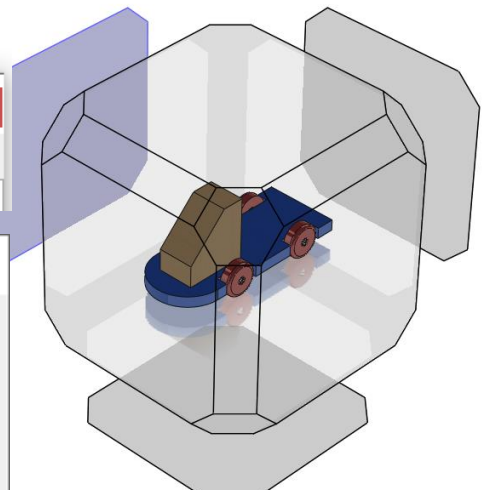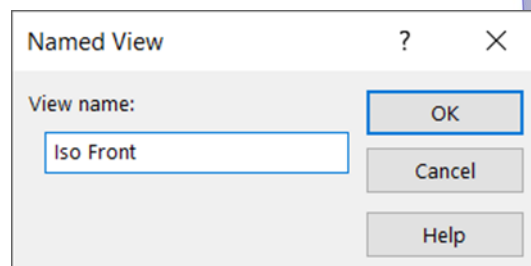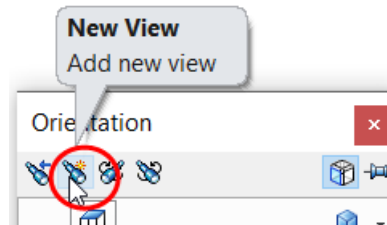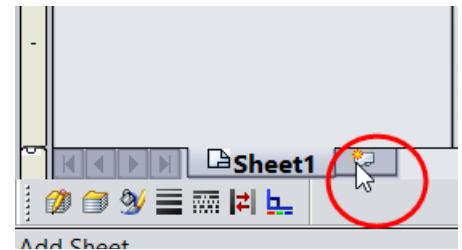Use the Note [A Note] command to add Text

Your Solution Drawing is now complete. Save your work

Detailed sheet

Select Add Sheet. To add an extra view, (isometric from the front:

1. Open the assembly (use R-Key for recent documents)
2. Select the required view using the view selector cube
3. Press the SPACEBAR and click **New View**
4. Name the view and select OK
5. Save the file
6. Open the drawing and in the view palette refresh the views
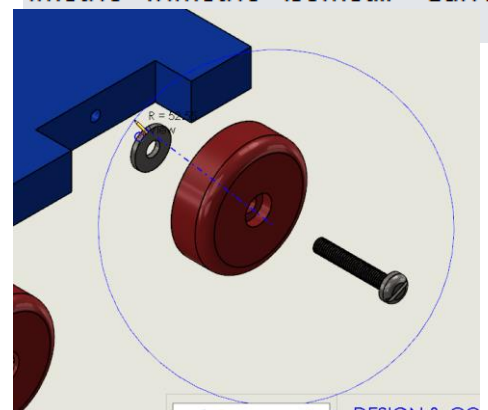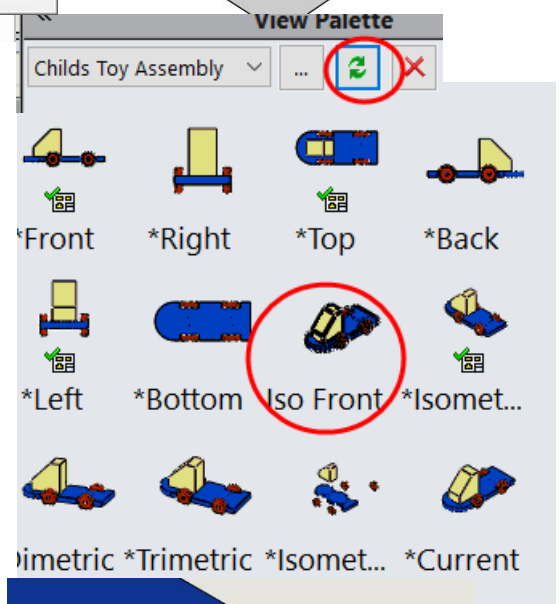7. Drag and drop the new view into the drawing

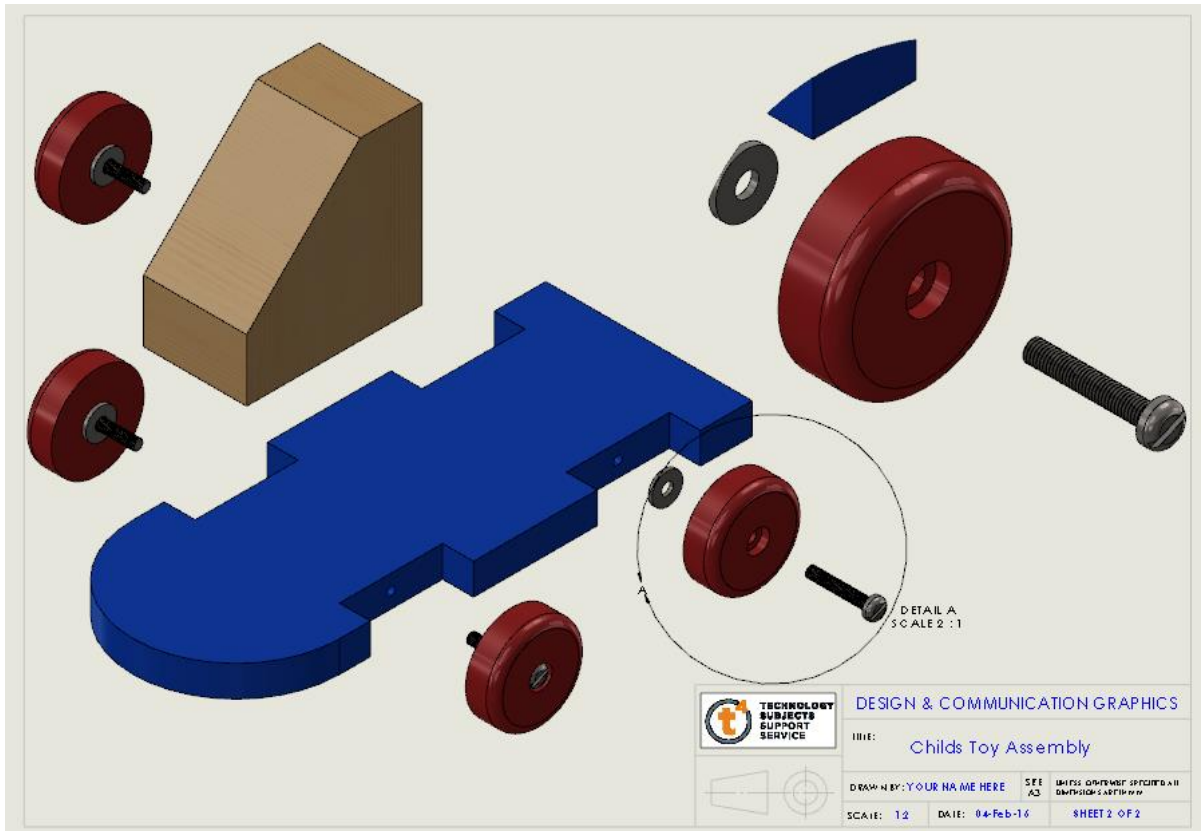Click on the view and select **Show in exploded state**

To explode
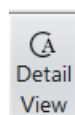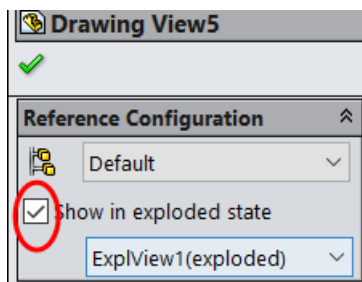
Add a detail view of the wheel

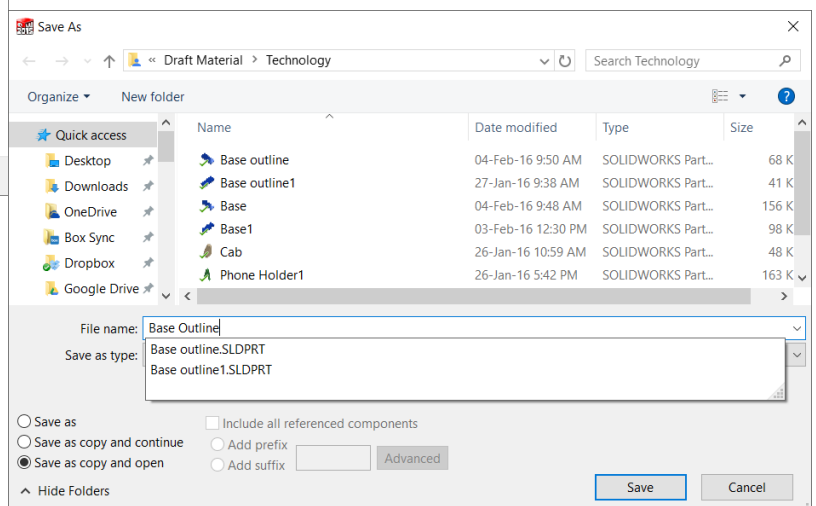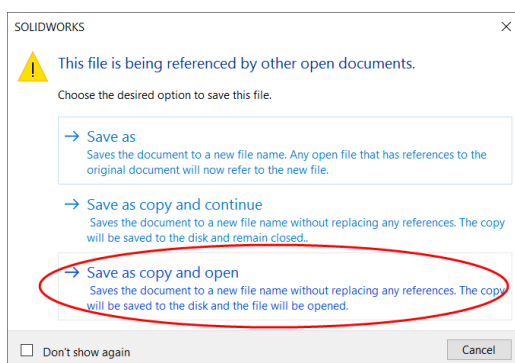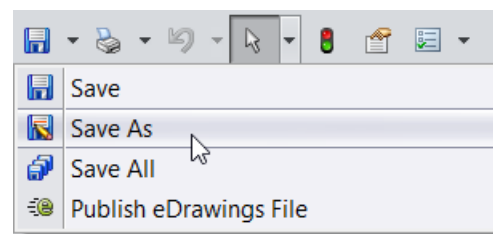Adjust the circle centre, radius and position of view as required

Creating worksheets

Open the **Base** part. Open **Save As.**

In the Design Tree select the features as shown
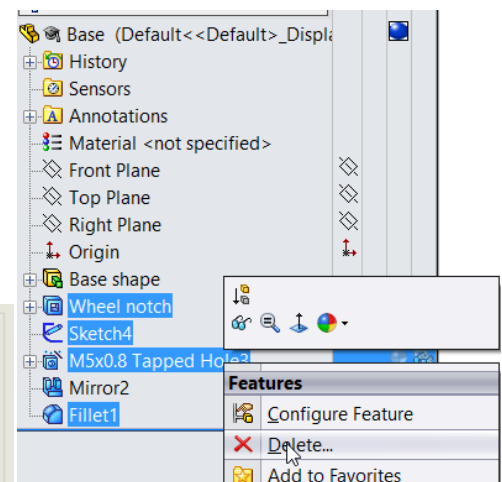
and **Delete.** Select **Save a copy and open.**
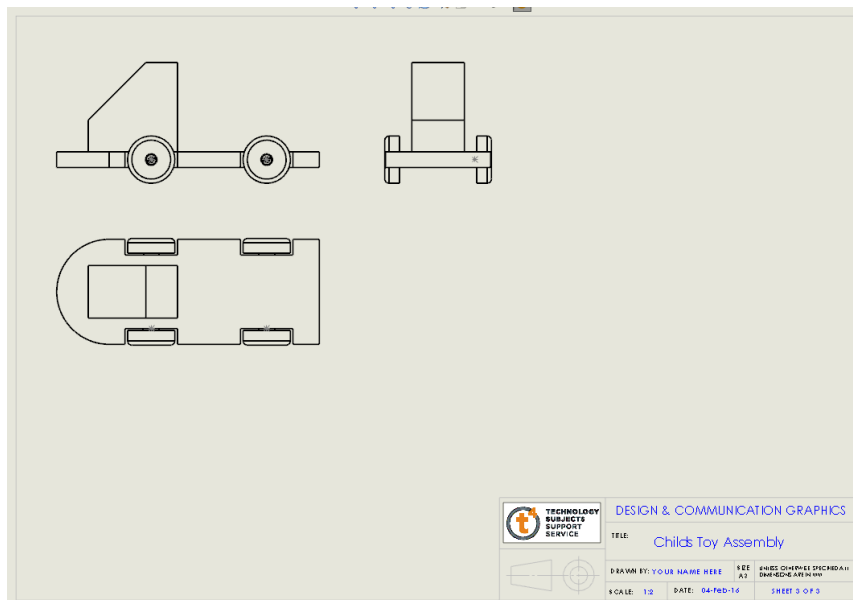

Save as **Base Outline**

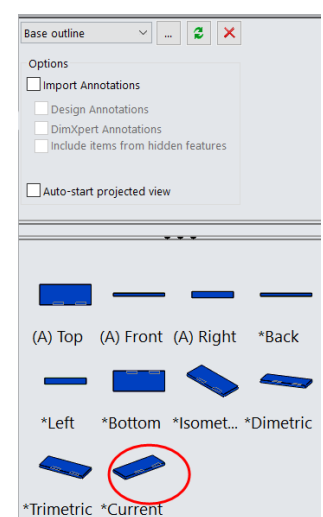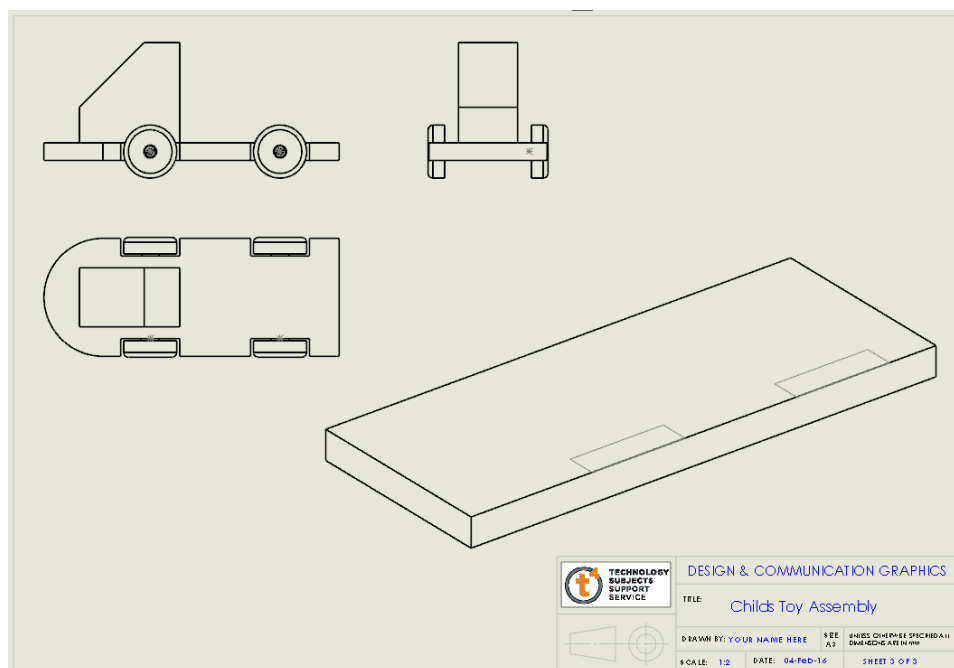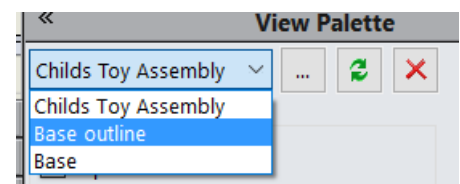In the copied part, delete the features as shown.

Save the part

Open the drawing (R-key)
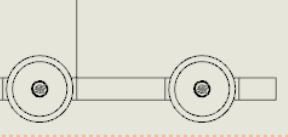
Add another sheet and add the orthographic views as before

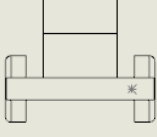In the **View Pallete** browse for the **Base outline** part.

Drag in the current isometric view

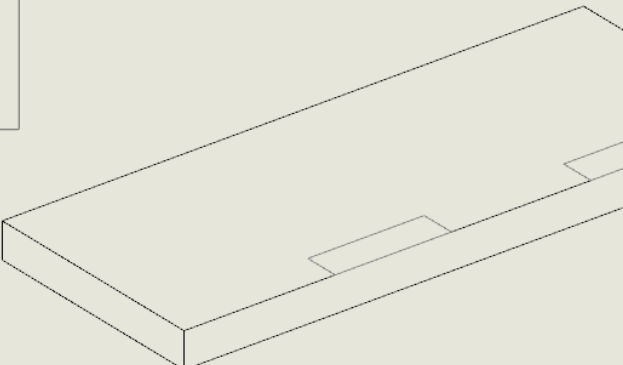Add appropriate notes to the worksheet and save

ELEVATION

END ELEVATION

PLAN

COMPLETE THE ISOMETRIC SKETCH OF THE CHILDS TOY

TECHNOLOGY
SUBJECTS
SUPPORT
SERVICE

DESIGN & COMMUNICATION GRAPHICS

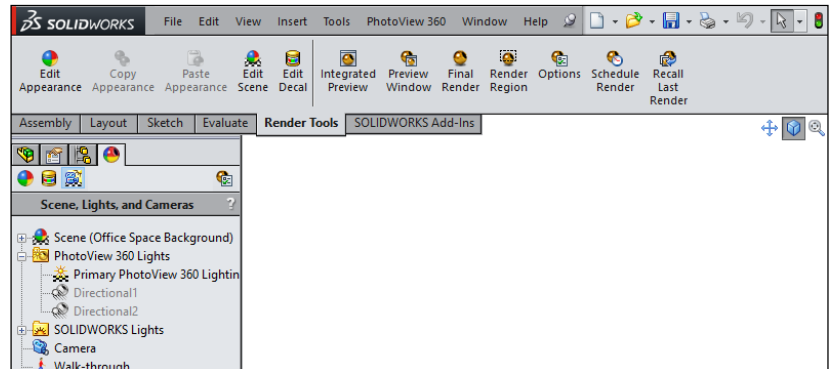TITLE: Childs Toy Assembly

DRAWN BY: YOUR NAME HERE

SEE A3

UNLESS OTHERWISE SPECIFIED ALL DIMENSIONS ARE IN mm

SCALE: 1:2   DATE: 04-Feb-16   SHEET 3 OF 3

PhotoView 360

PhotoView 360 is fully integrated

into SolidWorks

To add in render toolbar see

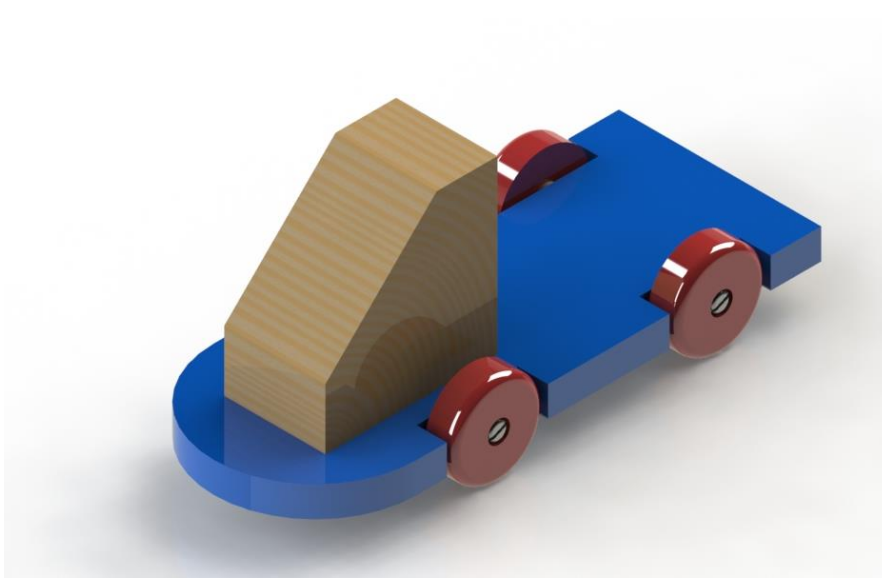http://t4.ie/sw/Photoview360.html

Open the **Childs Toy Assembly**. Select the render tools tab and **Render Region.** Adjust the

region to around the assembly
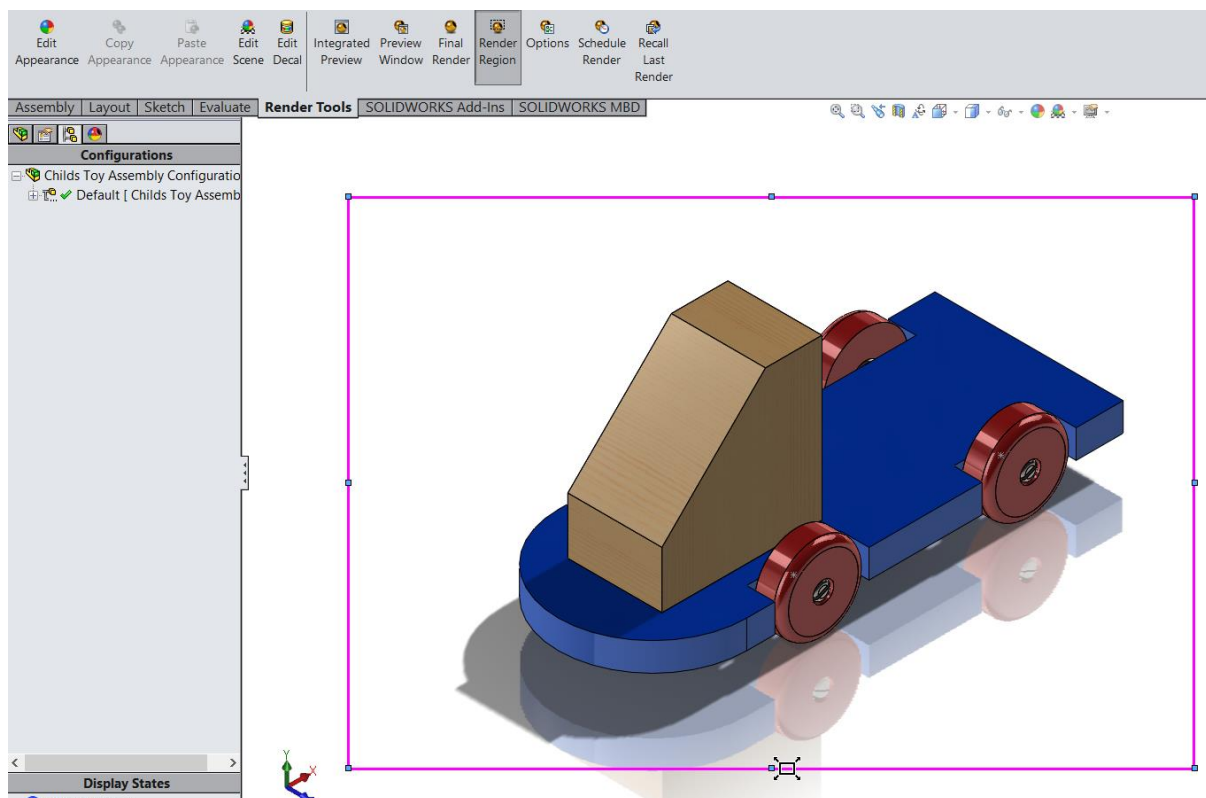
Select **Final Render** and allow PhotoView 360 to create the Rendered image. **Save Image** as

Childs Toy1

Exit PhotoView 360 and open the exploded view of the assembly. Render the image as

above and save as **Childs Toy2**