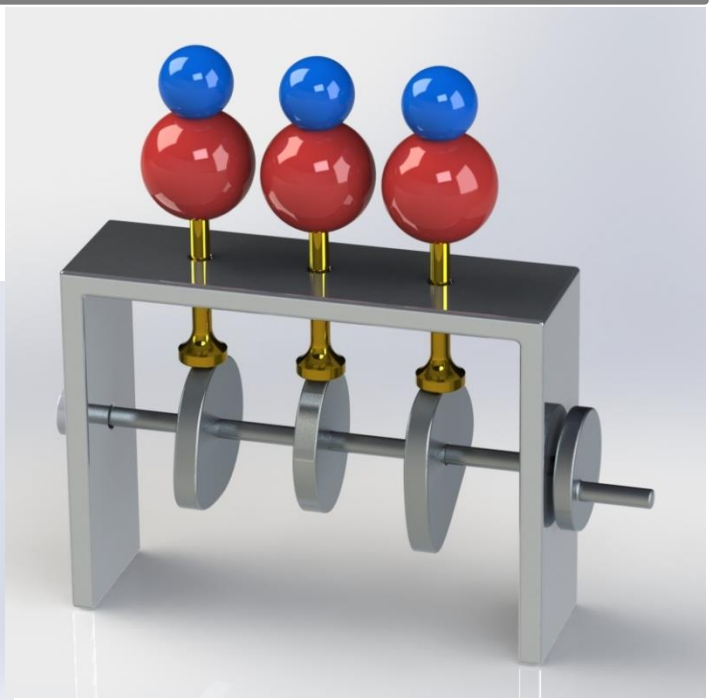
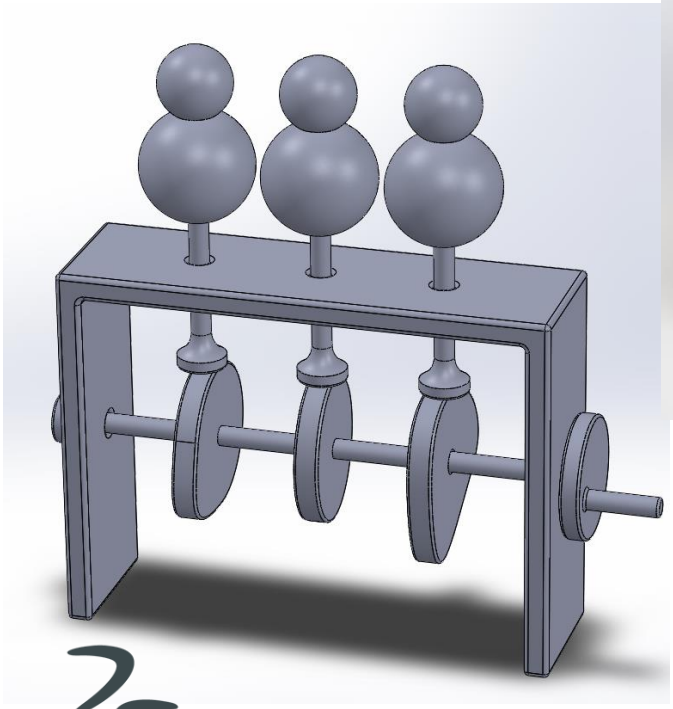
Solidworks/2014

3D Modelling Tutorial

L2





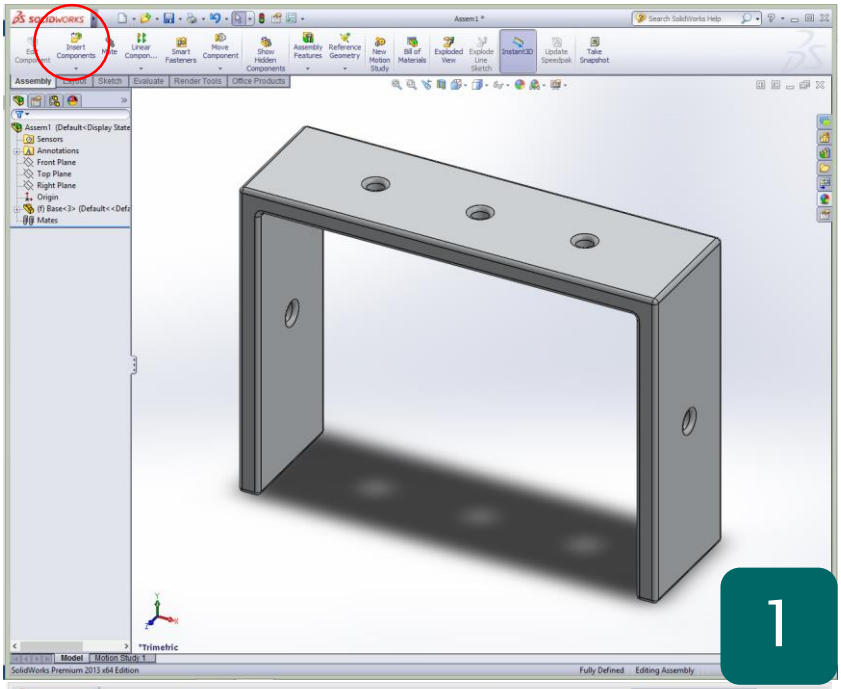
SOLIDWORKS

Learning Outcome; Cam Toy Assembly

Skill Level; 2 Intermediate

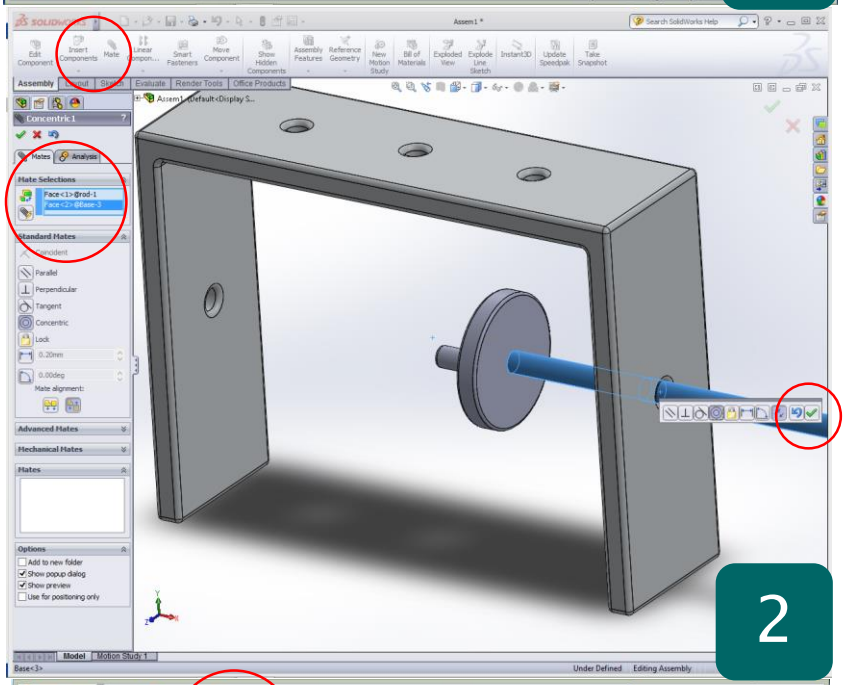
3D; *Assembly, Mate, Cam Mate, Lock, Motion Study, Render*

2D; *None required*



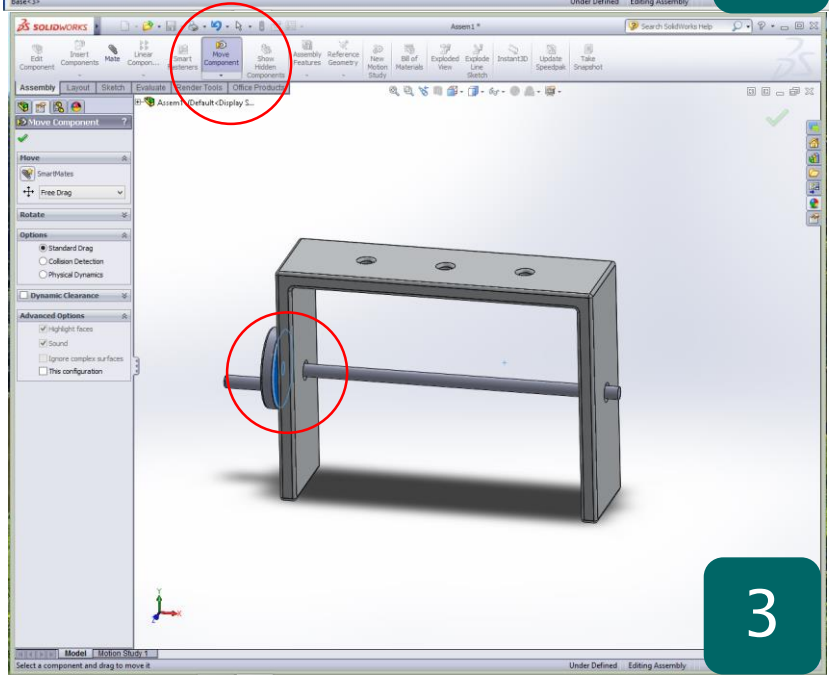
- Load a new Assembly document in Solidworks
- Click '**insert components'** and find the folder titled '**Cam Toy'**
- Shared > Design technology > Mr Billington > CAD Tutorials
- Click on '**Base'** and confirm

Why? – When creating an Assembly in Solidworks it is good practice to begin with the largest part as the building block to which all other parts are mated to



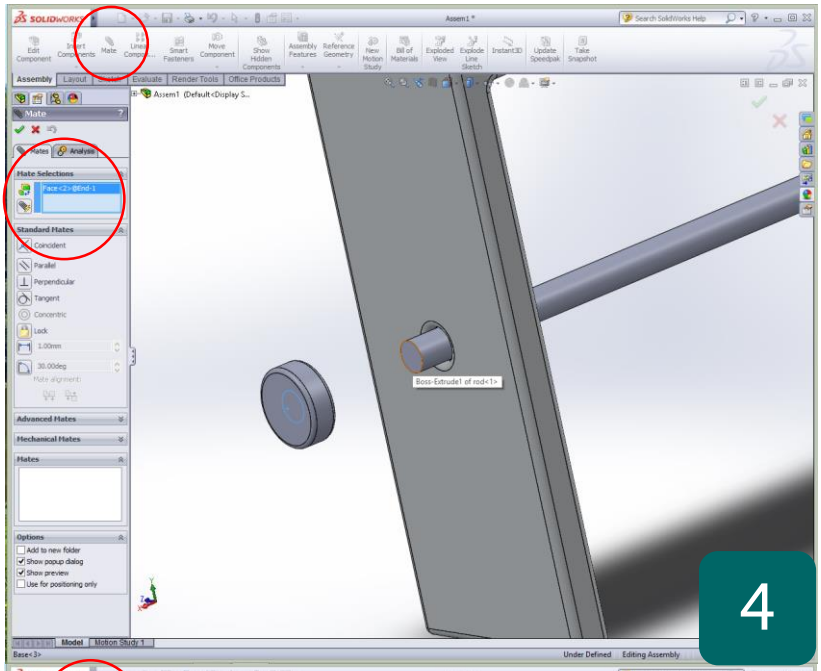
- Next insert **'Rod'** into the assembly from the same folder
- Click **'mate'** in the '**assembly toolbar'** and select the inside surface of the hole on the base side
- Click the outer cylindrical surface of the '**rod**'
- The two will align together with the software assuming a '**concentric**' mate
- Confirm with the green tick

Why? – Solidworks uses an intelligent mating process which decides on the most suitable 'mate relation' between two parts. As the two surfaces are cylindrical it makes sense to mate them via their centre axis



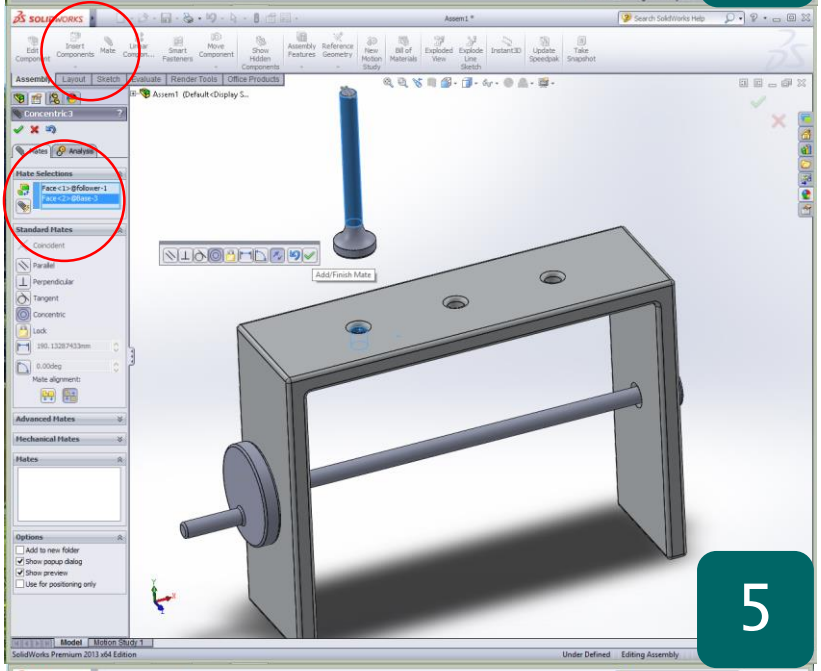
- Access the **'Assembly toolbar'** and click '**Move component'**
- Select a face on the '**rod'** to move from and drag the part through the **'base**' so that it looks roughly like the screenshot
- Confirm the placement with the green tick

Why? – In the real world a cam toy often has the rotating rod held loosely between the 'base' to allow for minimum friction when working. In its location it now becomes easier to assemble the other parts



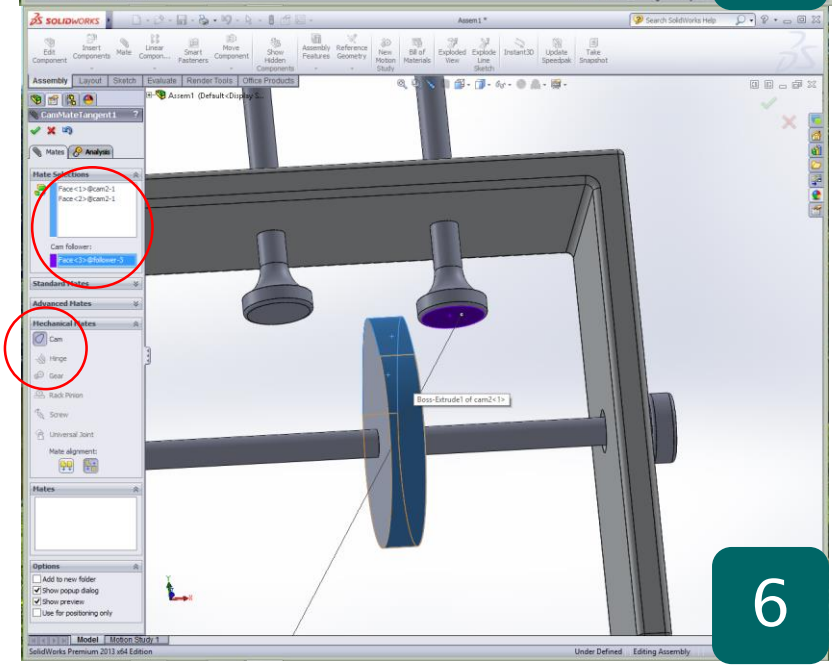
- '**Insert Components'** and access the part **'End'** and load into assembly
- Use the **'Mate'** tool to secure it to the end of the **'rod'**
- Inside cylindrical face of **'end'** > Outer cylindrical face **of 'rod'**
- bottom face of hole into **'end'** > End face of **'rod'**
- Confirm with the green tick

Why? – This end cap will prevent the 'rod' from moving through the base piece allowing for accurate animation and movement later on



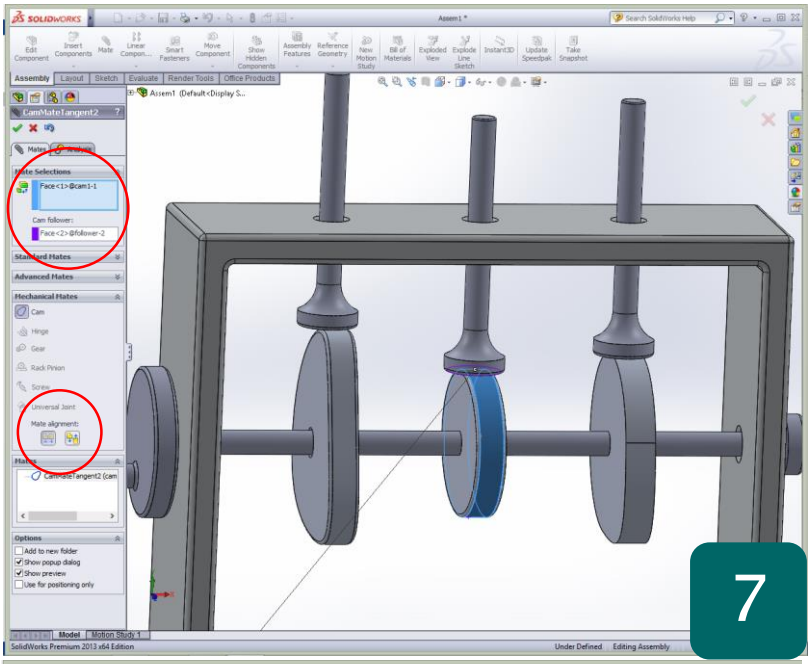
- **'Insert Components'** and load 'follower' into the assembly
- Use the 'mate' tool to make the two surfaces shown 'concentric'
- Use the 'move component' tool to locate it down into the base
- Repeat this process twice more until you have three '**follower'** components in each of the holes in the **'base'**

Why? – As this is an assembly with moving parts objects will only require 2 mates to allow them to move within the model. 3 mates should be used if it is to be fixed in place



- **'insert components'** and locate the first cam '**Cam1**' to load into the assembly
- '**Mate**' to place the cam '**concentric'**
- **'Move'** the component so that it sits roughly under the first **'follower'**
- Select **'mate'** from the '**assembly toolbar'**
- Find '**mechanical mates'** on the left feature menu and expand it clicking on the arrows
- Select '**Cam**' mate from the list
- Now select the outside face of the cam in the first '**profile**' box
- Select the 'cam follower' as the bottom face of the '**follower'**

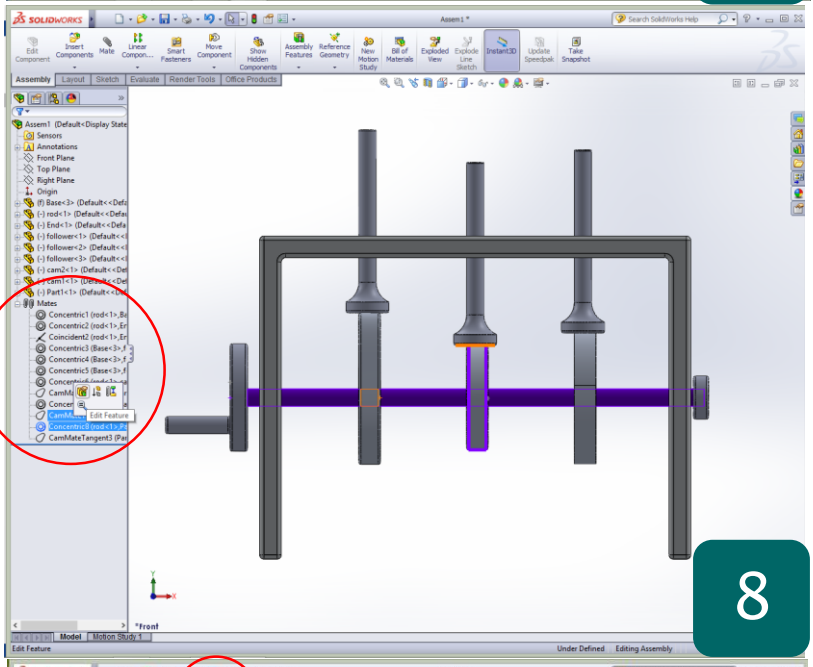
Why? – To make a part follow another in an irregular path is not possible without using this specially designed 'mate' that allows it to follow a path



- Repeat this process with the other two cam profiles; **'Cam2'** & **'Cam3'**
- Each time ensure the whole outer surface is selected as the profile
- The feature will only work if the path is complete without open ends

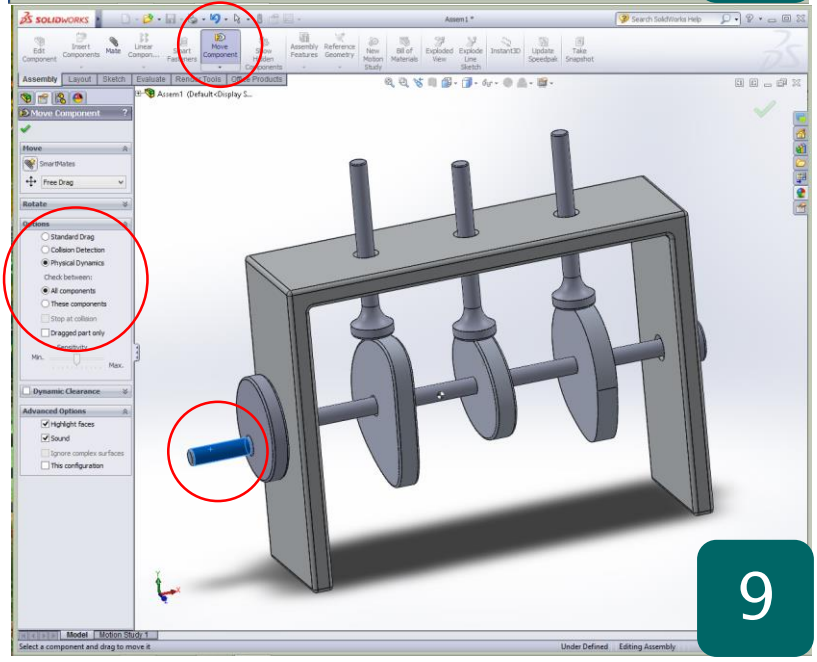
- *If the 'follower' drops to the base of the 'cam' simply click 'mate alignment' to the opposite direction*

Why? – 'Mate alignment' is the direction at which the mate is to be applied. Sometimes Solidworks can suggest the wrong option for this and it requires changing



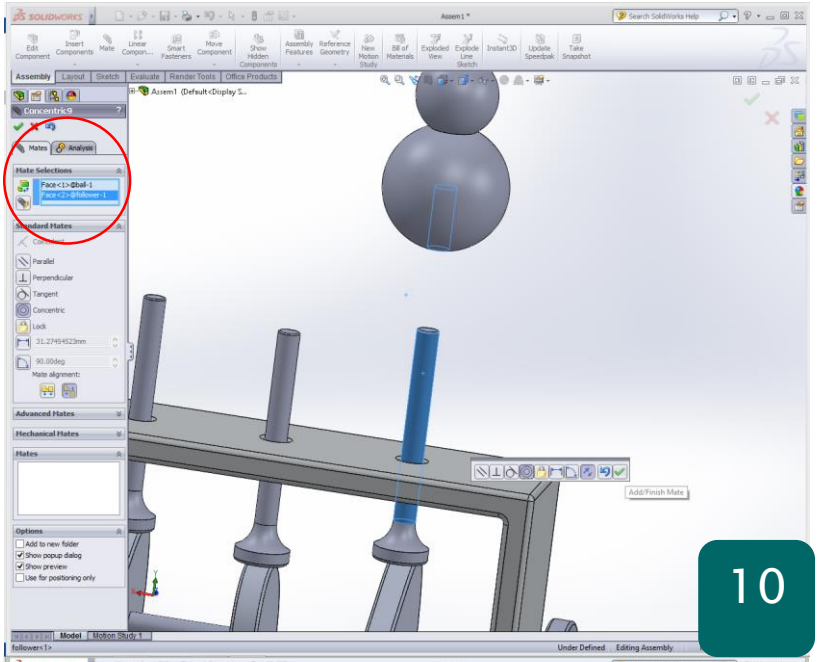
- Now the 'cams' are mated to the 'followers' they need fixing in place to the 'rod'
- Access the **'model tree'** on the left and click the **'+'** next to '**Mates'**
- This will load the assembly mate history
- Click on the '**concentric mate'** you used to fix the **'cam'** to the **'rod'**
- Select it and click **'Edit feature'** in the menu
- Now in the 'mate' feature menu click on the **'Lock'** (padlock)
- Confirm and repeat for each cam

Why? – In the real world the cams would be secured with a friction fit however since there is no friction in this assembly they must be locked in place to allow them to spin with the 'rod'



- Select the '**Move component'** feature from the **'assembly toolbar'**
- In the left feature menu select '**Collision Detection'**
- Now select the handle of the '**rod**' part and click to drag and rotate the component
- This should show the movement of the cam and follower pairings if mated correctly

Why? – If 'collision detection' or 'physical dynamics' is not selected when moving components Solidworks will not observe the mass of the objects and travel straight through them



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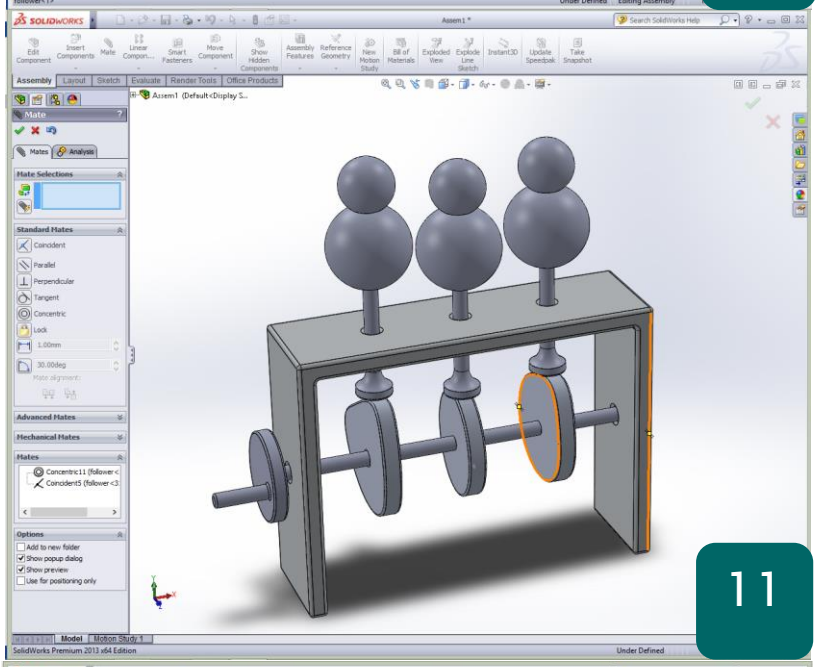
- **'insert components'** and load '**ball**' part into the assembly
- Mate using a '**concentric mate'** from both cylindrical edges
- Finally mate with the inside hole face on **'ball'** and the end face of '**follower**'
- Confirm with the green tick after each mate

Why? – The top part on the follower could be changed for any shape of design you require. A simple double ball shape was chosen due to spherical surfaces rendering very well due to the many light conditions across it.



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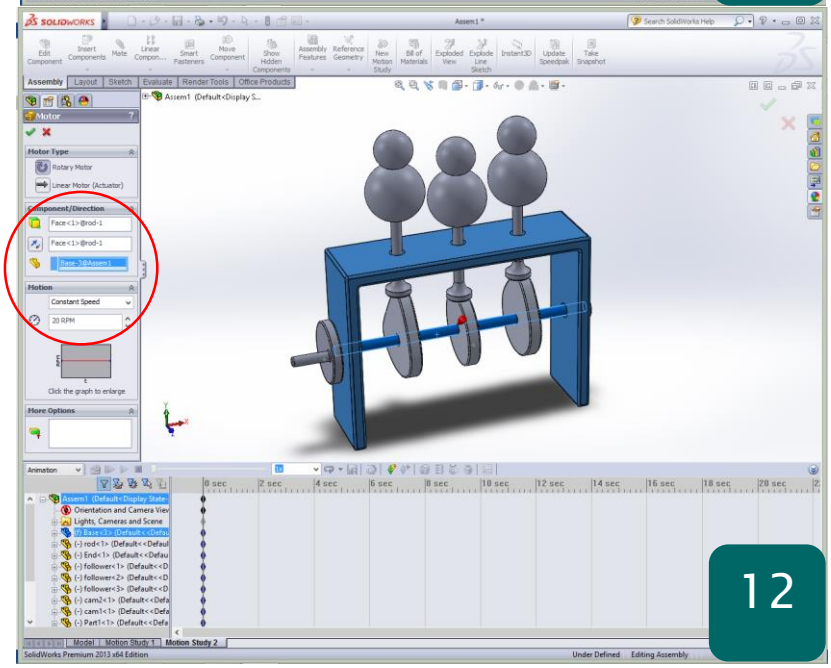
- When assembled the finished cam toy should look as shown with each '**ball'** figure on top of each **'follower'**
- Use **'Move component'** to test the rotation and movement of the cam toy
- Notice the different movement each figure makes as it follows the cam profiles
- These profiles can be edited to test different movements

Why? – Each cam profile can be edited through accessing the model tree and clicking on the cam and opening the part. Any changes made to the profile and saved will be updated in assembly



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- In the **'Assembly'** toolbar select '**Motion Study'**
- This will load up an animation bar below the model as shown
- Click the icon near the middle of the screen '**Motor'**
- Select the central shaft as the location of the motor
- Select the base as the reference point
- Set the **RPM to 20**
- Confirm with the green tick
- In the animation pane you can now click play to animate as though there is a motor on the middle shaft spinning it around

Why? – An animation study is designed to show how a 3D assembly would behave if moved in the real world. Many moving parts can be animate and save as a movie file to play in other media