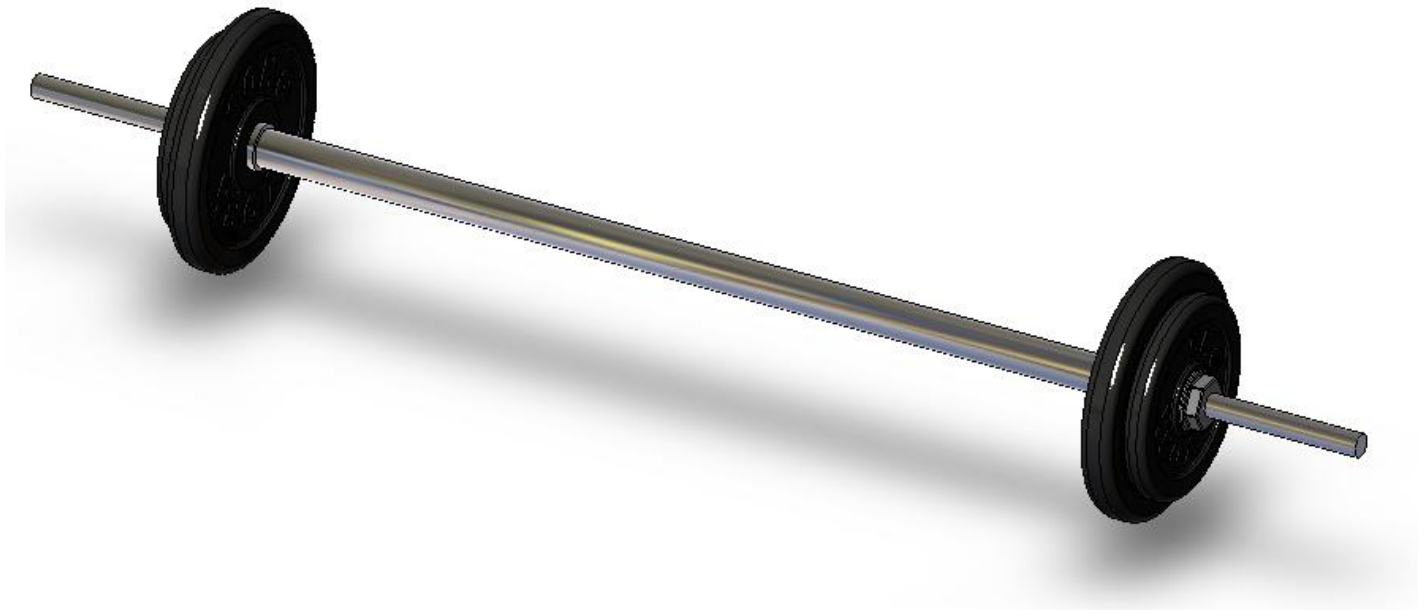# Weights Exercise – Introduction to Assemblies

Prerequisite Knowledge To complete this exercise you will need;

- to be familiar with the SolidWorks interface and the key commands.
- basic file management skills
- the ability to rotate views and select faces edges and surfaces.

Focus of the Lesson   On completion of this exercise you will have:

Created a new assembly
Saved an assembly
Added components; through browsing and dragging from the feature manager tree.
Rotated Components
Applied Coincident, Concentric and Parallel Mates between surfaces and, surfaces and planes
Deleted Mates

Commands Used   *Make Assembly from Part/Assembly, Insert Component, Rotate Component, Mate.*

Creating the Assembly

Bottom-Up Assembly   **Bottom–Up Assemblies** are created by adding and orientating existing parts in an assembly. Parts added to the assembly appear as **Component Parts.** Component parts are orientated and positioned in the assembly using **Mates.** Mates relate faces and edges of component parts to planes and other faces/edges.

Stages in the process

Creating a new assembly
New assemblies are created using a similar method as new parts

Adding the first component
Components may be dragged and dropped from an open window or selected from a standard browser.

Position of the first component
The initial component added to an assembly is automatically fixed as it is added. Other components may be repositioned after they are added.

Feature Manager Design Tree and Symbols
The Feature Manager includes many symbols which contain information about the assembly and the components in it.

Mating components to each other
Mates are used to position and orientate components with reference to each other. Mates remove degrees of freedom from the components

Make assembly from Part/Assembly   Use the **Make Assembly from Part/Assembly** option to generate a new assembly from an open part. The part is used as the first component in the new assembly and is fixed in space.

Where to find it.   Click **Make Assembly from Part/Assembly** 🔲 on the standard toolbar
Or, Select **File, Make Assembly from Part**

Getting Started   The files for this exercise are pre-prepared and located in the folder
4 - Introduction to Assemblies 2009/Dumbell assembly Tech. Resource CD
or the T4 website
Copy the entire folder from the CD to your hard drive or memory key

Open an existing part   Open the part **Chrome Bar.** A new assembly will be created using this part
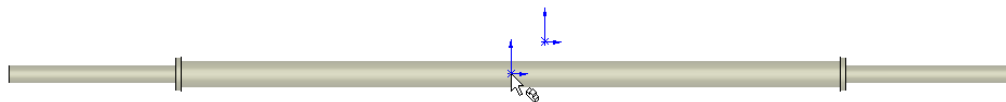
Click **Make Assembly from Part/Assembly** 🔲

Choose the default assembly template.
Click **OK**
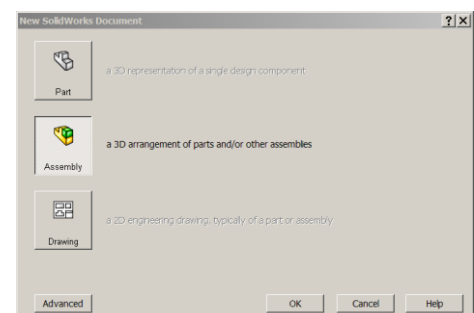
Inserting the first part The **Insert Component** Dialog box appears with **Chrome Bar** displayed. Ensure **graphics preview** is selected

Move the cursor into the drawing area. A preview of the part along with the origins are displayed

Move the cursor to the origin and select, the part origin will snap to it as displayed.

Choose **Trimetric View**

Saving the Assembly Select **File, Save as** on the standard toolbar.
Save the assembly as **Weights & Bar** into the folder containing the parts used to create it. An assembly is identified by its extension *.sldasm*.

Assemblies need to reference the parts used to create them. Should you wish to share an assembly, it is essential that the parts used to create it accompany the file.

Components Parts that are inserted into the assembly appear in the Feature Manager Design Tree and may be expanded to show the individual features of that part

State of the Component The part may be fully, over or under defined. A (+) or (-) sign will precede the part name if it is **Over** or **Under Defined.** Parts that are under defined have some degrees of freedom available. **Fully defined** have none.
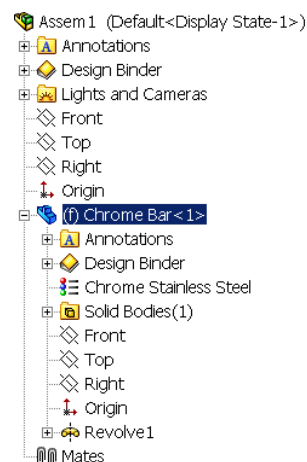
Mates **Mate Group:** All **Mates** in an assembly are placed in a folder, identified by a double paper clip icon in the feature manager tree.

Mates may be used to fully define a component that does not move, or under define a component that is intended to move.
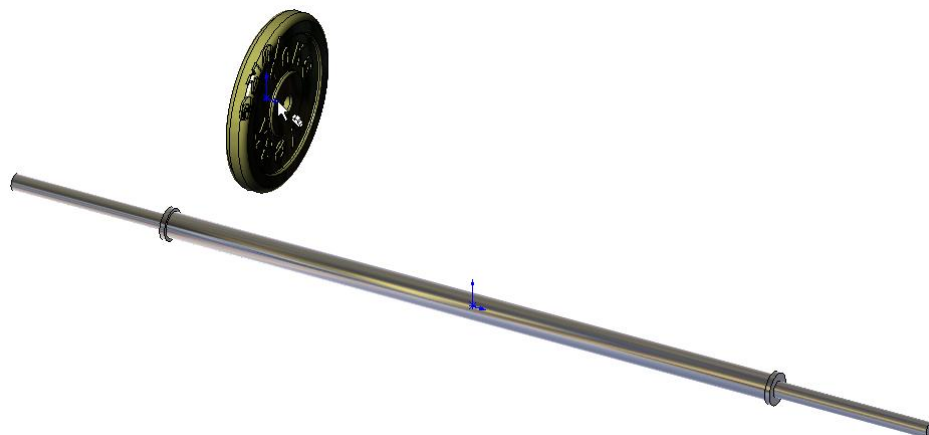
Adding Components Select **Insert Component** [Insert Comp...] from the **Assembly Toolbar**

Choose **Browse** from the **Insert Component dialog box**

Choose **10Kg WEIGHT** from the folder of parts. Choose **Open**

A preview is displayed in the drawing area.
Click to drop it as shown overleaf.

Leaving Certificate Technology

Moving Components	Holding down the left hand mouse button on the component will allow you to drag it around
Rotating Components	Select **Rotate Components** [Rotate Comp...] from the **Assembly Toolbar.**
	Place the rotate symbol over the component, hold down the left hand mouse button and drag. The component will rotate through its available degrees of freedom.

This is not to be confused with Rotate View [icon] from the View toolbar.

In order to create mates it is essential that we are proficient at rotating views of parts, in order to select faces/edges.

|	
Insert Mate	**Insert Mates** creates relationships between component parts or between parts and an assembly.
Where to find it.	Choose **Insert, Mate…** Or Select **Mate** [Mate] from the **Assembly Toolbar**
Adding the Mates	Select **Mate,** [Mate] the **Mate Property Manager** will appear.
Selecting Faces	Select the face of the weight. Rotate the view and choose the face of the shoulder of the bar.
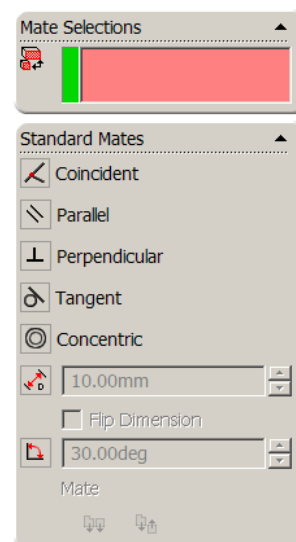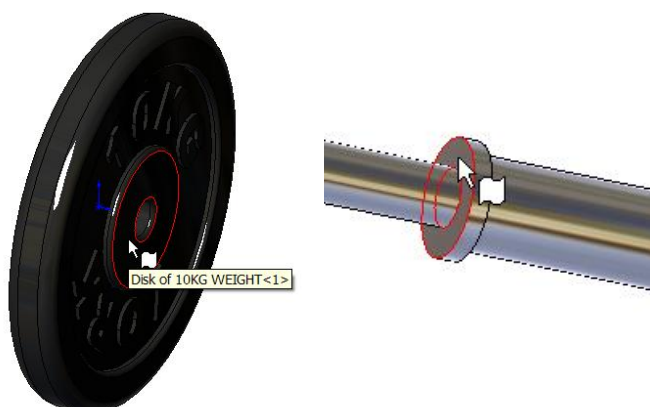
The parts will move so that the selected faces are contained on a single plane.

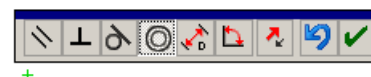A **Coincident Mate** will be chosen by default

Choose OK

A Coincident Mate has been applied.
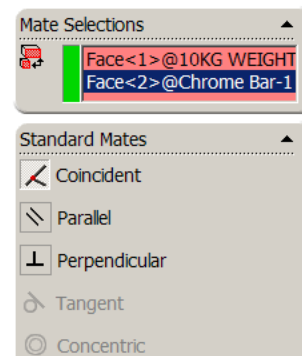
Mate Pop-up Toolbar

The **Mate Pop-up Toolbar** is used to make selections easier by displaying the available mate types on the screen.
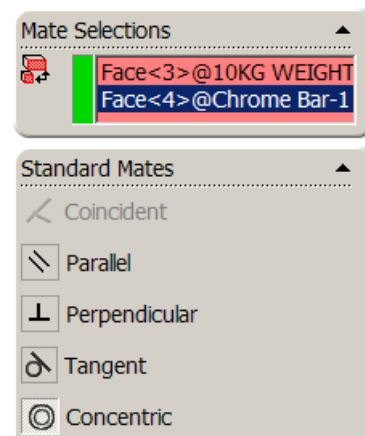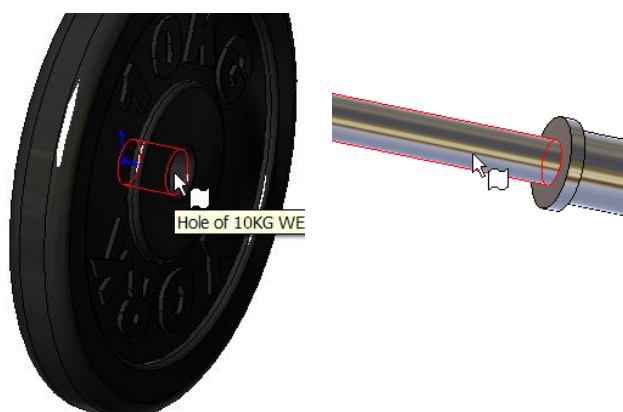These mirror those that appear in the property manager.

Further Mates

The **Mate Properties Manager** will remain open on screen awaiting the next component parts to which mates will be applied.

Select the surface of the bar and the internal surface of the hole on the weight.

Concentric Mate is selected by default.
The weight will move so that its axis becomes concentric with the axis of the bar.

Choose OK   to apply the mate.

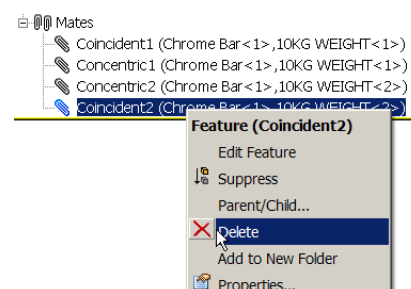Choose OK   to exit the property manager.

Deleting Mates
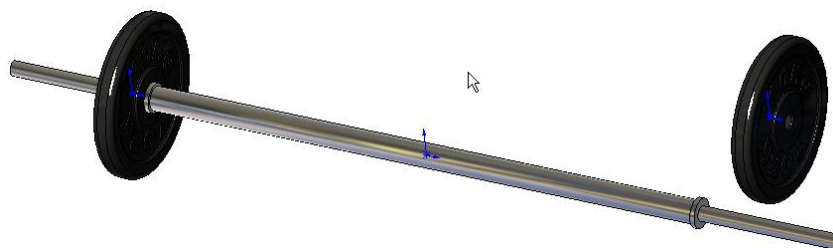
To delete a mate between two components;

Double click on the **Mates Folder** at the bottom of the **Feature Manager Design Tree.**

All mates will be displayed.

Right click on the desired mate and choose **Delete**

Adding further parts   Insert the part **10KG WEIGHT** as before. Position as shown.

Adding Mates   Add a **Coincident Mate** between the left hand face of the weight and the face of the shoulder of the bar.
Add a **Concentric Mate** between the surface of the smaller diameter of the bar and the internal surface of the hole in the weight.
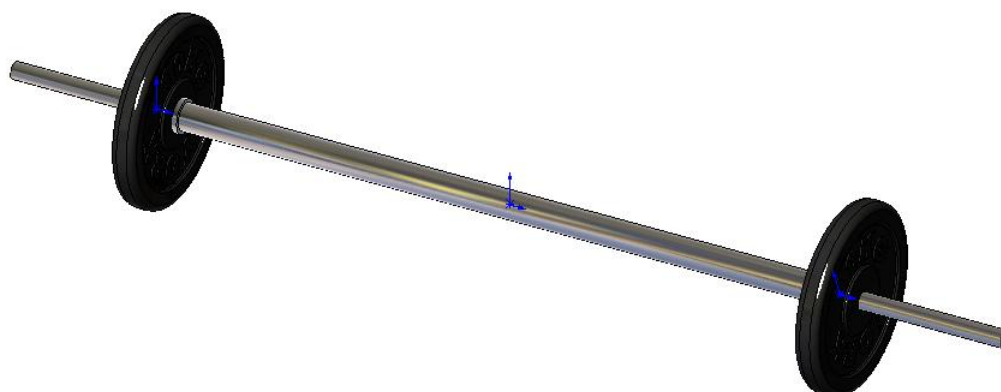
**Inserting the 5Kg Weight**   Insert the part **5KG WEIGHT.**

Adding Mates   Add a **Concentric Mate** between the surface of the smaller diameter of the bar and the internal surface of the hole in the weight.

The weight is now restricted to travelling only along the axis of the bar.

Create a **Coincident Mate** between the two faces of the 5Kg and 10Kg Weights.

The 5Kg Weight will move into position as shown.
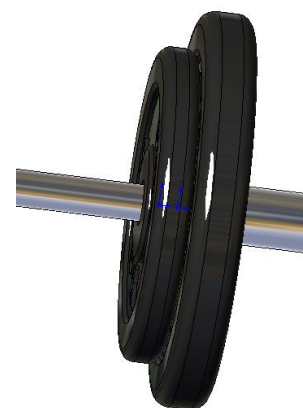
Adding another 5KG WEIGHT   To add in another **5KG WEIGHT**, hold down the **Ctrl** key, select and drag the **5KG WEIGHT** from the **Feature Manager Tree**. Drop it into the drawing area.

The inserted weight will have the same orientation as the existing weight in the assembly.

Position the 5kg weight on the opposite side of the bar using **Coincident and Concentric Mates** between the necessary faces as before.
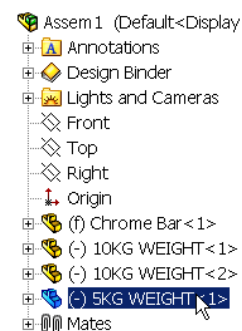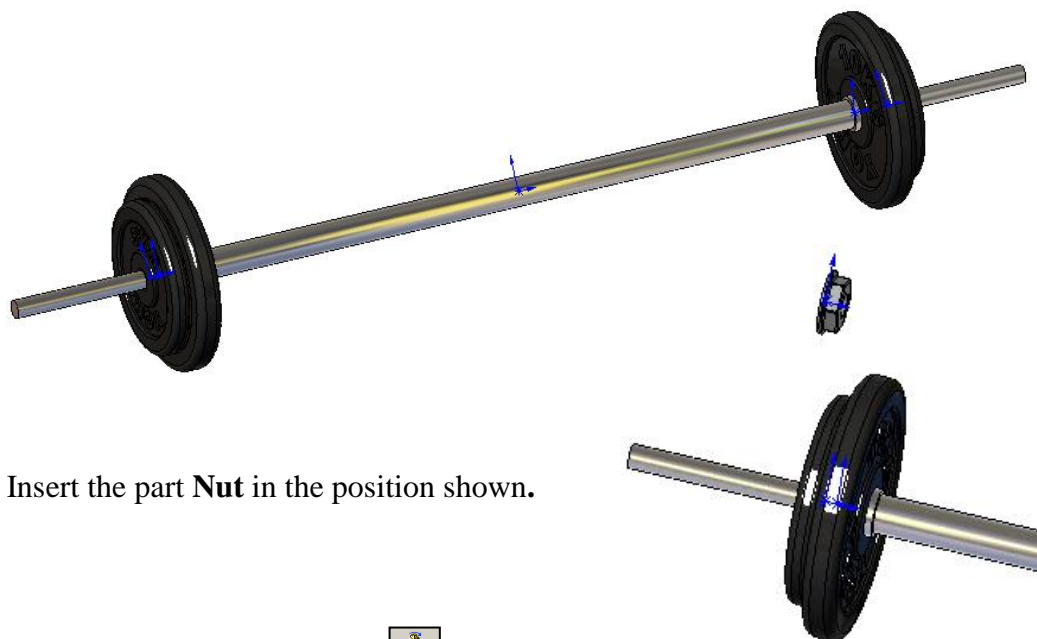
Adding the Nut   Insert the part **Nut** in the position shown**.**

Rotating Components   Select **Rotate Components** [Rotate Comp...] from the **Assembly Toolbar.**

Place the rotate symbol over the component, hold down the left hand mouse
button and drag.

Rotate the nut through 180º to the position shown.

Applying Mates   Apply a **Concentric Mate** between the surface of
the bar and the internal surface of the hole in the nut.

Apply a **Coincident Mate** between the face of the nut
and the face of the 5Kg weight.

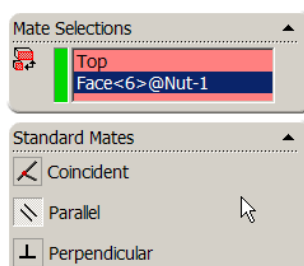
Preventing rotation   The nut is still free to rotate on the bar.

To prevent the nut from rotating a **Parallel Mate** must be added between one of
the faces of the nut and the **Top Plane.**

Choose any one of the faces of the nut.
Expand the feature manager design tree
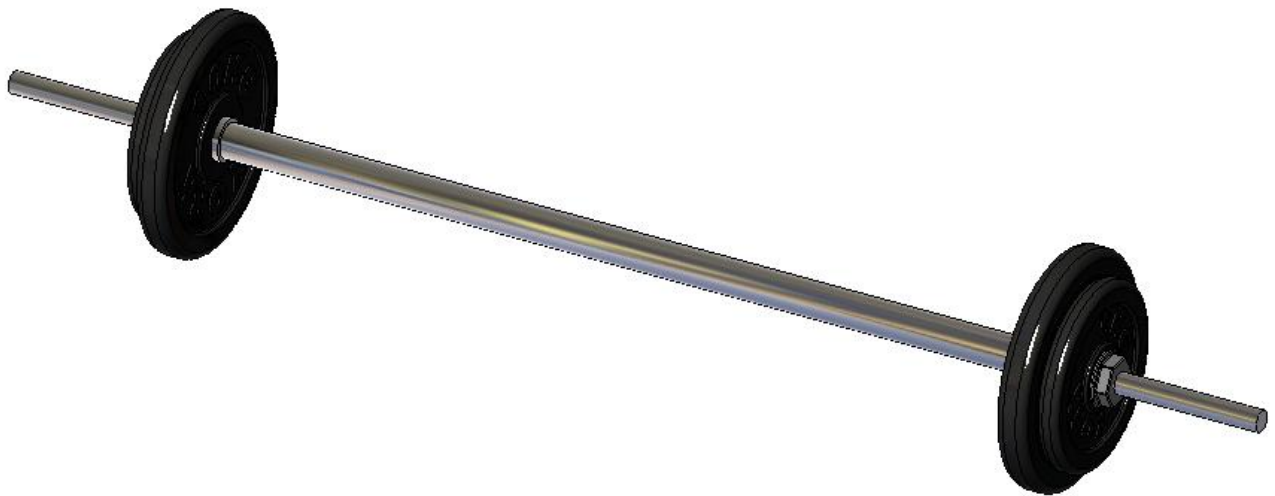in the drawing area and choose the top
plane.
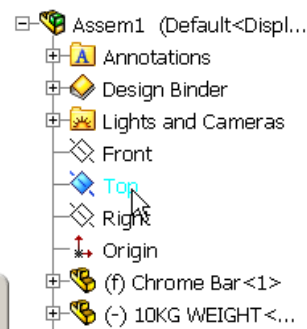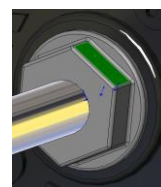
Apply a **Parallel Mate** between
the surface and the plane.

Adding another Nut

Holding down the **Ctrl key,** drag another nut from the feature manager design tree.

Position using a **Coincident Mate** between the faces, and a **Concentric Mate** between the hole and the surface of the bar.

Add a **Parallel Mate** between one of the faces of the nut and the top plane, to prevent rotation.

Exercise Complete!