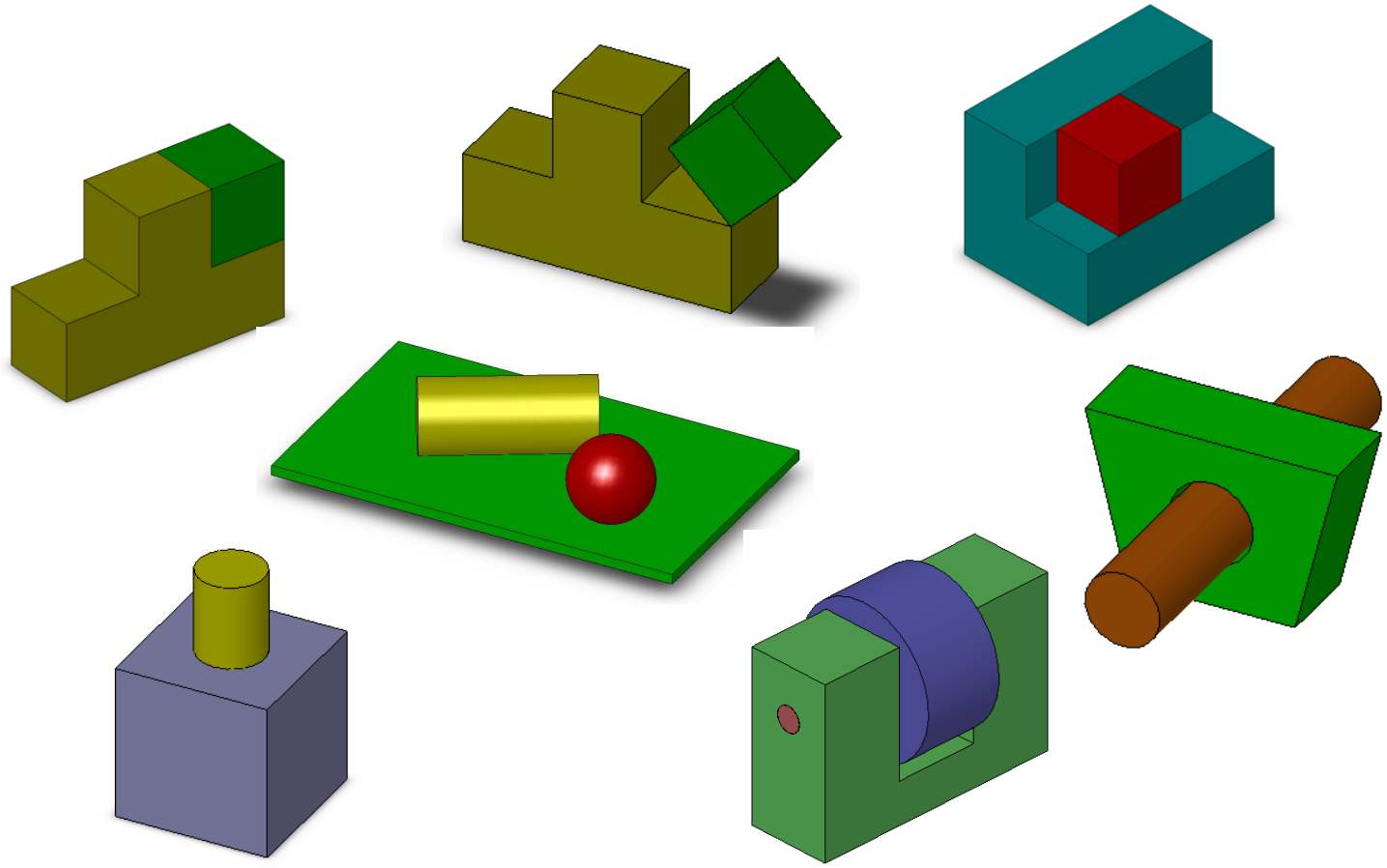
Introduction to Assemblies – Concept Mates



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 number of simple assemblies
Saved the assemblies
Added components through browsing Windows Explorer
Moved & Rotated Components
Applied Coincident, Concentric, Angled, Tangent and Distance Mates between surfaces and, surfaces and planes.
Fully Defined parts.
Deleted Mates

Commands Used ***Make Assembly from Part/Assembly, Insert Component, Rotate Component, Move Component, Show/Hide Planes, Float, Mate.***

Creating an 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.

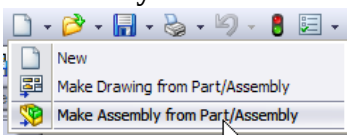
Mating components to each other
Mates are used to position and orientate components with reference to each other. Mates remove degrees of freedom from the components

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 **Page icon** and **Make Assembly from Part/Assembly**
or, Select **File, Make Assembly from Part**



Getting Started

The files for this exercise are pre-prepared and located in folder 4 – Introduction to Assemblies 2009/Introduction to Assemblies – Concept mates on the Technology resource CD

Copy the entire SolidWorks folder from the CD to your hard drive or memory key

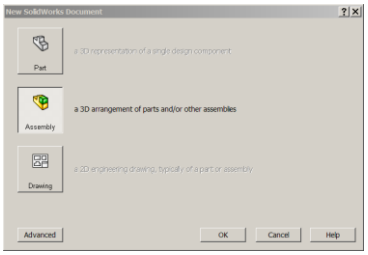
Coincident & Angled Mate

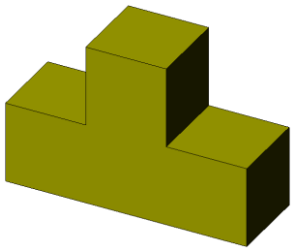
Open an existing part

Open the part **SolidWorks/Assemblies/Mates/Coincident Mates/T Block**
A new assembly will be created using this part

Click Page icon and **Make Assembly from Part/Assembly**
Choose the default assembly template.

Click **OK**

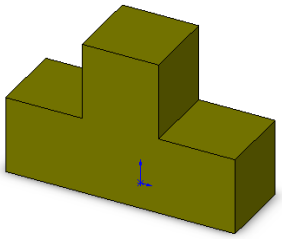


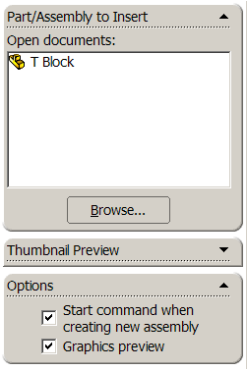


Inserting the first part

The **Insert Component** Dialog box appears with **T Block** displayed.
Expand **Thumbnail Preview.** Ensure **graphics preview** is selected.

Move the cursor into the drawing area.
A transparent preview of the part along with both the part origin and the assembly origin are displayed.





Move the cursor to the origin and select. The part origin will snap to the assembly origin.

If a three dimensional view is not displayed, choose **Trimetric View**

Saving the Assembly

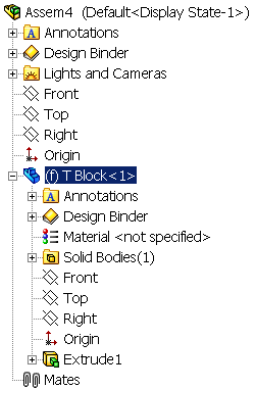
Select **File, Save as** on the standard toolbar.
Save the assembly as **Block Assembly** into the folder containing the parts used to create it. An assembly is identified by its extension **.sldasm*.

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 assembly. Hence an assembly and its component parts are stored in a folder.

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.



Fixed position

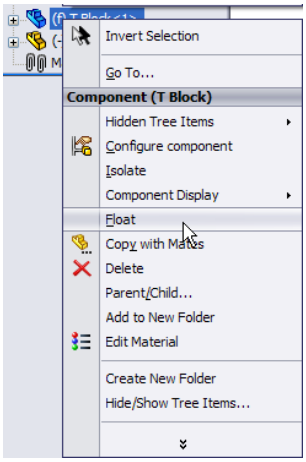
The **(f)** preceding T Block indicates that it is **fixed** in position. Should you try to drag it, it will not move. The first part inserted into an assembly document is **fixed** by default.

Floating a part

Floating a part returns it to an under defined state **(-)**.

To **float** a part that has been **fixed**; Right-click the component in the graphics area, or the component's name in the Feature Manager design tree.

Select **Float** from the pop-up menu.



Moving Components

Holding down the left hand mouse button on the component will now allow you to move it by dragging.

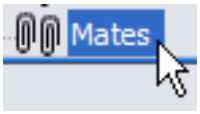
The part is no longer **Fixed.**

Mates

Mates may be used to fully define a component that does not move, or under define a component that is intended to move.

Mates may be added between faces, edges, points, planes etc

Mate Group: All **Mates** in an assembly are placed in a folder, identified by a double paper clip icon in the feature manager tree.



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**  from the **Assembly Toolbar**

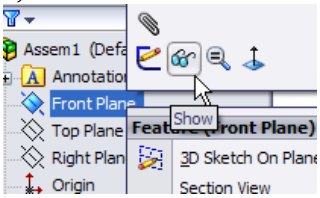
Mating Surfaces and Planes

For this exercise we will mate the T Block with the Front, Right and Top Planes**.**

Display Planes

To begin we must make the 3 planes visible.

Right click on the Front Plane in the Feature Manager design tree and select **Show**. 



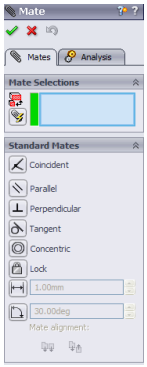
The plane will be visible in the drawing area.
Repeat the procedure for the Right & Top Planes.

Command Manager

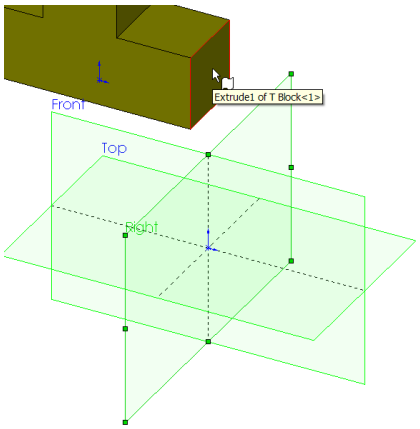
Choose **Assemblies** from the **Command Manager**
The **Assembly Toolbar** is displayed

Adding the Mates

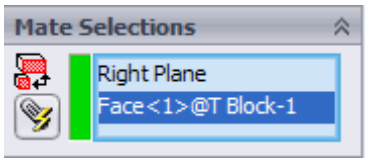
Select **Mate,**  the **Mate Property Manager** will appear.
The **Mate Property Manager** displays the standard mates.
Only those suitable for the geometry selected will be available.



The remainder will be greyed out.
Select the **Right Plane** and the right face of the block, as shown overleaf.



These will appear in the **mate selections** window.



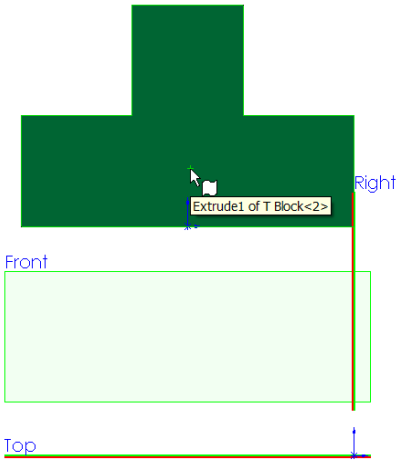
Coincident Mate will be chosen by default and the part will move to enable the face to become coincident with the right plane.

Choose **OK**

A different mate type may be selected from those available should you so require.

Moving the part

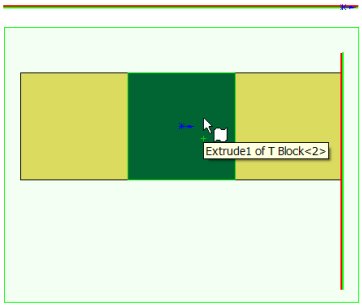
Choose **Front View**



Hold down the left mouse button on the part, and drag.

The part is free to travel vertically but constrained to travel along the right plane. It has lost some of its **degrees of freedom**

Choose **Top View**

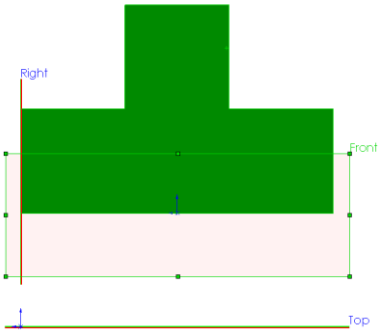


The part is free to travel horizontally but constrained to travel along the right plane.

Further Constraint.

Ensure that no faces are selected
Select **Back View.**

Choose **Mate** from the **Assembly Toolbar.**



Select the **Front Plane** and the **back face** as shown.

Coinicident Mate will be chosen by default.

Choose **OK**

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.

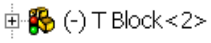
Moving the part

Choose **Top View**

Hold down the left mouse key on the part and drag. The part is no longer free to move in this direction, this degree of freedom has been removed by the coincident mate to the front plane.

Fully Defined

In the Feature Manager design tree the *minus* still precedes the part name.

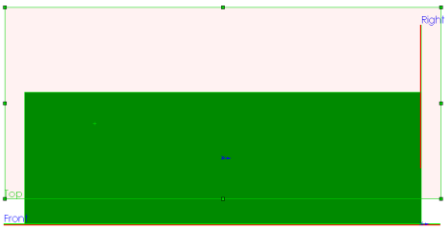


This indicates that it is not yet fully defined

Further Mate

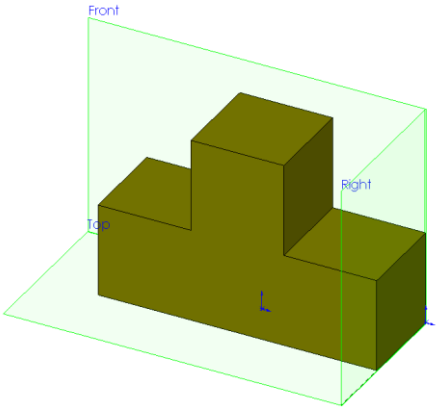
Choose **Bottom View**

Apply a **Coincident Mate** between the base surface and the **Top Plane**



Choose **Trimetric View.**

The part may no longer be moved by dragging.
All degrees of freedom have been removed.
The part is now fully defined

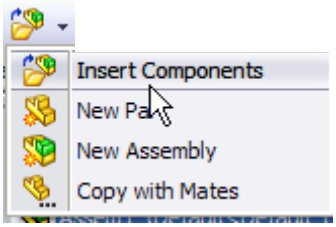


Hiding Planes

To **Hide** a plane, right click on the plane and choose **Hide.** Hide all planes.

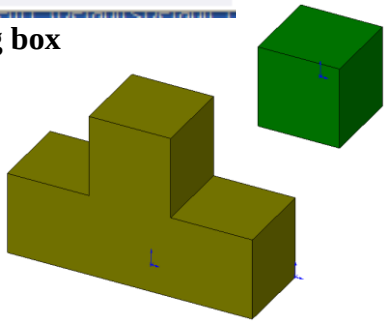
Adding Components

Select **Insert Component** from the **Assembly Toolbar**



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

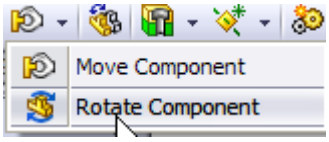
Choose **Square Block** from the folder of parts.
Choose **Open**
A preview is displayed in the drawing area.



Click to drop it as shown.
This part is not **fixed**. It is free to move and rotate.

Rotating Components

Select **Rotate Components** from the **Assembly Toolbar.**



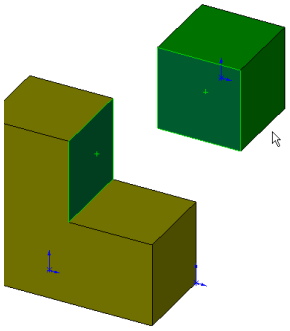
Place the rotate symbol over the component, hold down the left hand mouse button and drag. The component will rotate through its available degrees of freedom.

This is not to be confused with Rotate View 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.

Coincident Mate

Select **Mate,** the **Mate Property Manager** will appear

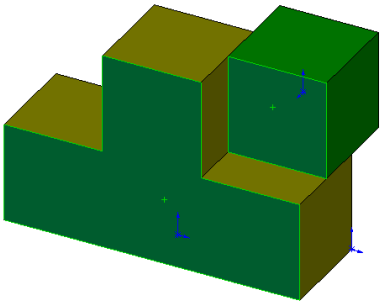


Select the faces shown opposite.
Create a **Coincident Mate** between the faces

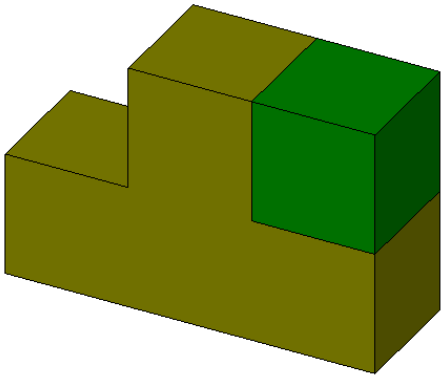
Create a further **Coincident Mate** between the underside of the block and the top face of the step

Fully Defined

The block is still free to slide along the mated faces

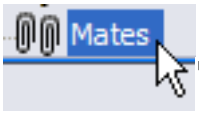


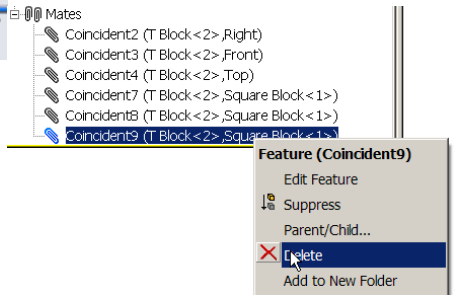
To fully define the part a coincident Mate must be added between the faces displayed.



Deleting Mates

To delete a mate;
Double Click the paper clip icon in the Feature Manager design tree
All mates within the assembly will be displayed.





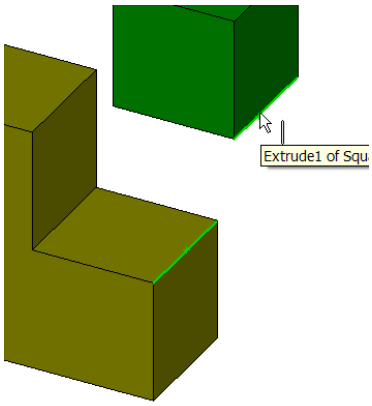
Right Click on any mate and select **Delete.**
This will restore a **degree a freedom** to the part.

Delete the 3 Coincident Mates between the T Block and the Square Block. The Square Block is free to move through all degrees of freedom again.

Mates between edges

A **Coincident Mate** may also be created between **edges**

Select **Mate,**  the **Mate Property Manager** will appear.



Choose the two edges highlighted opposite.

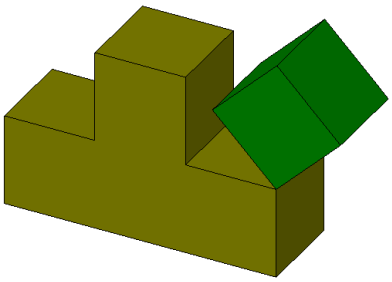
Choose **Coincident Mate.**

Add a further **Coincident Mate** between the front faces of both blocks.

Choose **OK**  **,** and **OK**  again.

Moving the part

Drag the part.

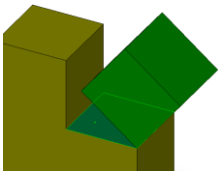


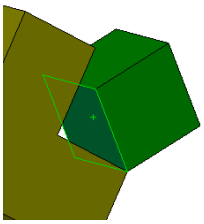
Because of the mates selected the part rotates around the mated edge.

It is not necessary to fully define all components within an assembly. Under defined parts may be used to display motion in an assembly.

Angled Mate

Select **Mate.**
Choose the surfaces shown opposite.





Coincident Mate is selected by default.

Choose **Angled Mate**.

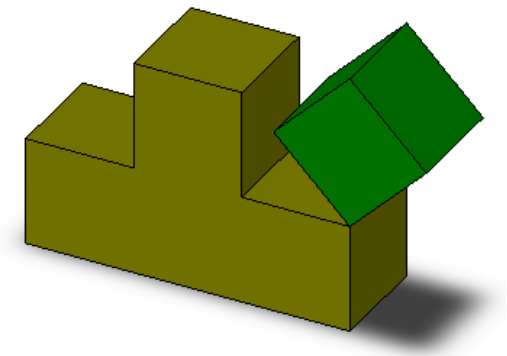
Input a value of **45º.**

Investigate the use of **Aligned/Anti-Aligned**

Select **OK** 

Fully Defined

Drag the block. The part no longer rotates around the coincident edge. It has no **degree of freedom** available. It is **fully defined.**



Save and Close Save and Close the assembly.

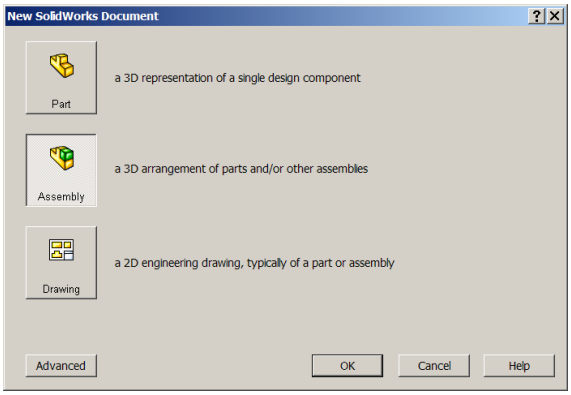
Distance Mate

Getting Started Choose **File, New…**

Select the default assembly.

Choose **OK**

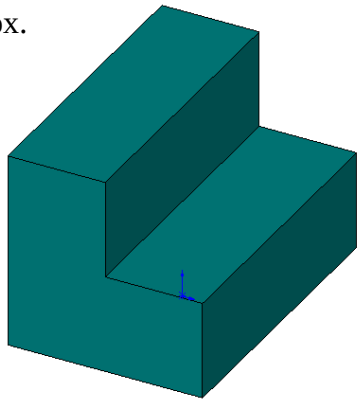
An assembly may be generated by creating a new assembly document from the dialog box shown and then subsequently inserting the first part.



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

Browse to;
SolidWorks/Assemblies/Mates/Distance Mates

Select the part **L Block** and drop it fixed to the origin.

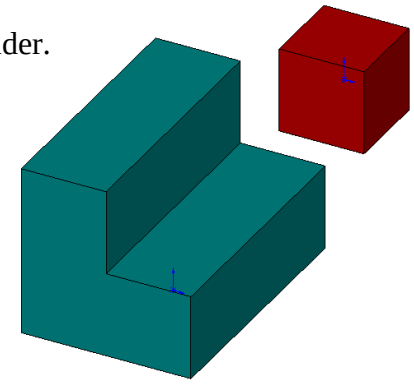


Save the Assembly **Save** the assembly as **Distance Mates** in the **Distance Mates folder**

Further Components Choose **Insert Component** from the **Assemblies Toolbar**

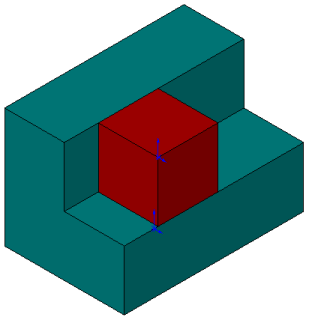
Select **Square Block** from the Distance Mates folder.

Drop it into position as shown.



Concentric Mates Add **Concentric Mates** between the vertical and horizontal faces to constrain the square block as shown below.

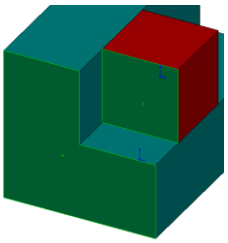
Degrees of Freedom Drag the block. It still has the freedom to slide along the faces with which it is mated.



Distance Mate

Used to define a distance or gap between parts, faces, edges or points.

Select **Mate** 



Choosing faces

Choose the highlighted faces shown opposite

Choose **Distance Mate**
Enter a **distance** value of **30mm**

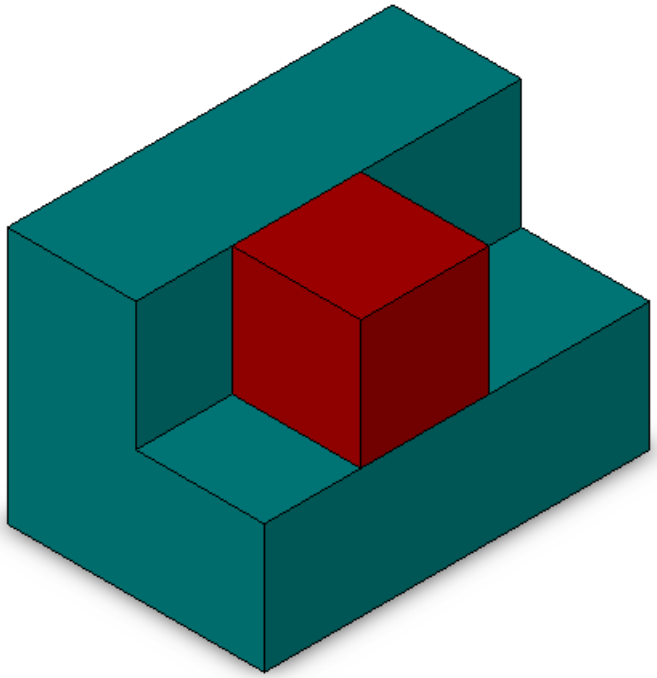
The block will move so that the distances between the faces is 30mm.

Fully Defined

It is no longer possible to drag the part. It is **Fully Defined**

Save & Close

Save & Close the assembly.



Concentric Mate

Getting Started

Create an assembly using the part **Cube** located in;
SolidWorks/Assemblies/Mates/ Concentric Mates

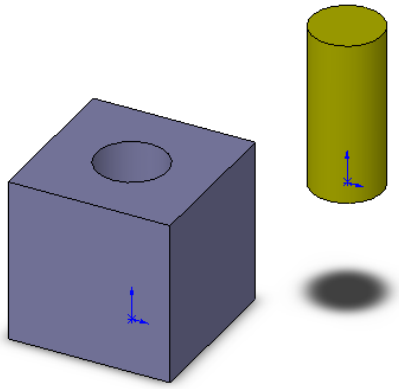
Fix the **part origin** to the **assembly origin.**

Saving the assembly

Save the assembly as **Concentric Mates** in the **Concentric Mates folder.**

Insert Components

Insert the part **Dowel** from the same folder.



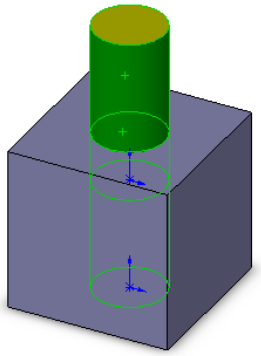
Concentric Mates

Select **Mates**
Choose the surface of the dowel and the internal surface of the hole.

Because of the geometry selected, **Concentric Mate** is displayed by default.

The dowel moves so that its axis coincides with the axis of the hole.

Choose **OK**

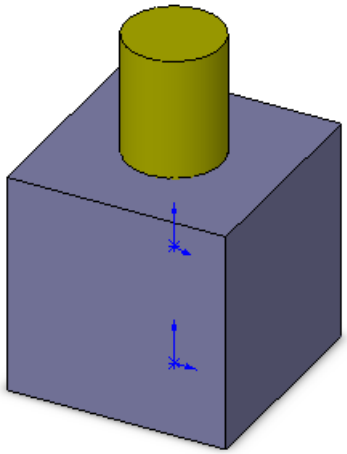


Move the part

Drag the part. Because it is **under-defined,** it will move within its remaining degrees of freedom – vertically along the axis of the hole and dowel.

Fully Defined

How would you completely restrict the movement of the dowel?

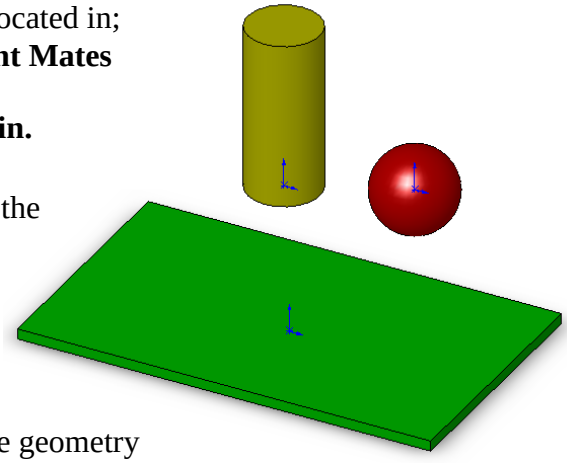


Tangent Mate

Getting Started

Create an assembly using the part **Base** located in; **SolidWorks/Assemblies/Mates/ Tangent Mates**

Fix the **part origin** to the **assembly origin.**



Saving the assembly

Save the assembly as **Tangent Mates** in the **Tangent Mates folder.**

Insert Components

Insert the parts **Dowel** and **Ball** from the same folder.

Tangent Mate

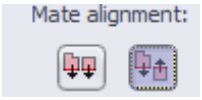
Tangent Mate is only available when the geometry selected may be made tangential to one another.

Choose **Mate**. Choose the top face of the dowel and the top face of the base. Tangential Mate is unavailable - greyed out. Choose X

Choose **Mate**. Choose the cylindrical face of the dowel and the top face of the base. **Tangential Mate** is now chosen by default. The dowel moves such that its surface is tangential to the top surface of the base.

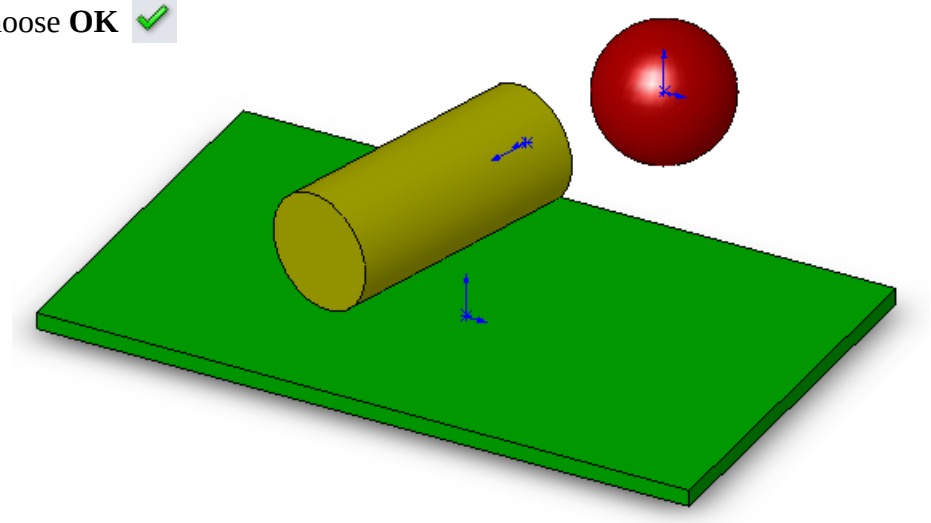
Aligned/Anti-Aligned

The dowel may be tangential to the underside of the surface.



Toggle between **Aligned** & **Anti-Aligned** and note the effect.

Choose **OK**



Move the dowel

Drag the dowel. The dowel will remain tangentially in contact with a plane containing the top surface at all times.

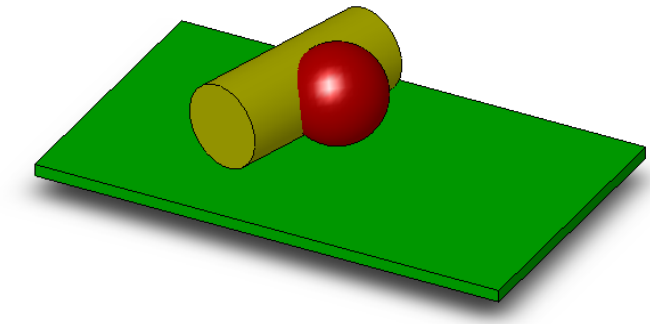
Further Mates

Add a **Tangent Mate** between the ball and the surface of the base. Investigate the effect of **Aligned** and **Anti-Aligned.**

Choose **OK** 

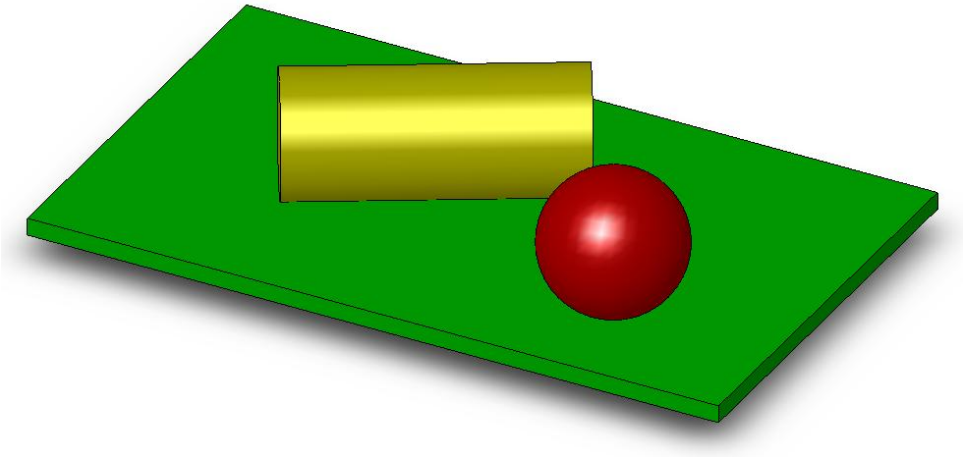
Problem

When the ball is dragged against the dowel it intersects it as shown below.



Solution?

At this stage, what could we use to solve this problem?



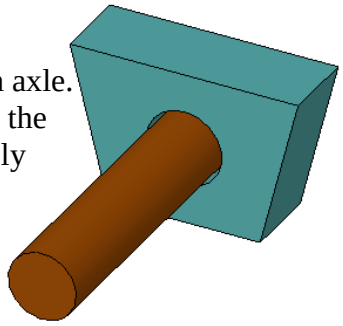
Width Mate

Width Mate

A **Width Mate** centres two **tab faces** within the width of two **reference faces**.

Getting Started

Open the assembly named **Width Mate** from the folder;
SolidWorks/Assemblies/Mates/ Width Mate



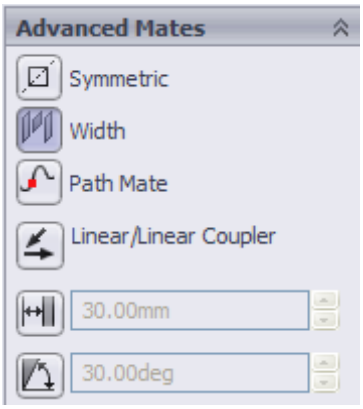
This assembly consists of an axle housing along with an axle. A concentric mate has been added between the axle and the hole. However the axle is still free to travel longitudinally along the hole axis.

The challenge is to centre the housing between the faces of the axle.

Knowing the length of the axle and the width of the housing, distance mate could be used. However, if dimensional changes were made to either part the housing would no longer be centred.

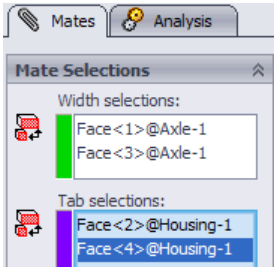
Width Mate

Select **Mate.**
Expand **Advanced Mates** and select **Width Mate.**

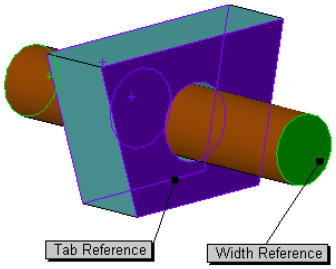


Select the faces of the axle as the **Width Selection**

Select the faces of the housing as the **Tab Selection**



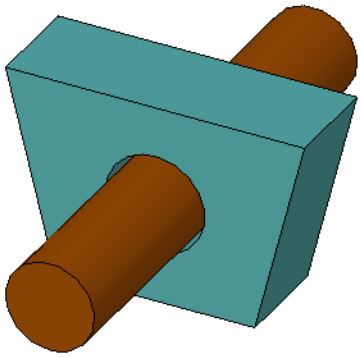
The axle moves such that the housing is centred on it.



Should any dimensional changes take place the housing will always remain centred on the axle.

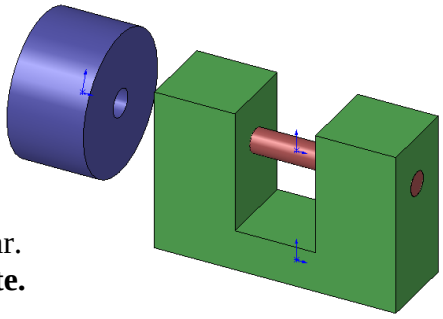
Save & Close

Save & Close the assembly.



Further Exercise

Open the assembly named **Width Mate 1** from the folder; **SolidWorks/Assemblies/Mates/ Width Mate1**

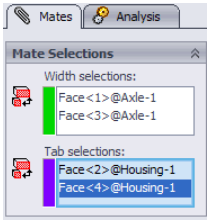


A concentric mate has been applied between the **sleeve** and the **swivel bar**. The sleeve is still free to travel along the axis of the bar.

We wish to constrain the sleeve such that it is centred between the internal faces of the U Bar. This will be achieved by adding a **Width Mate.**

Width Mate

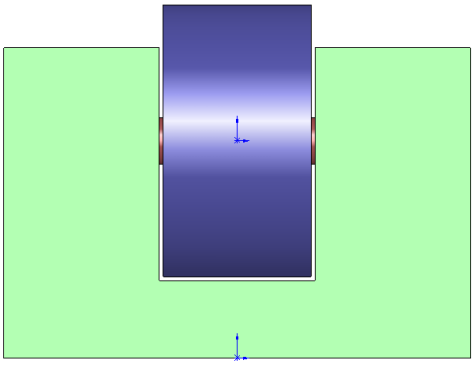
Select **Mate.** Choose **Width Mate.**



Select the internal faces of the U Bar as the **Width Selection**

Select the faces of the Sleeve as the **Tab Selection**

The sleeve will move to a position, centred between the two internal faces of the U Bar.



Drag the part

Drag the sleeve, it is still free to rotate around the swivel bar. It is not **fully defined**.