Solidworks/2014

3D Modelling Practice

3DS
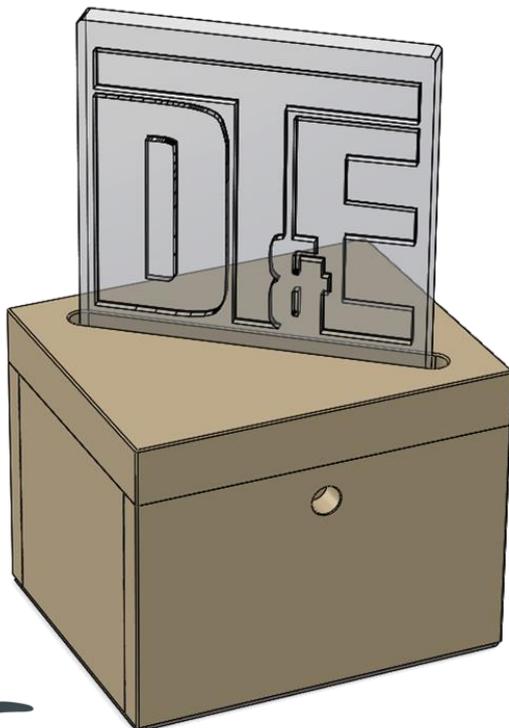
## SOLIDWORKS

Learning Outcome; Mood Light Assembly

Skill Level; 1 - Beginner

3D; *Assembly; Mate – faces, lines, planes, Render- Parts, Faces*

2D; *None Applicable in Assembly*

DT&E
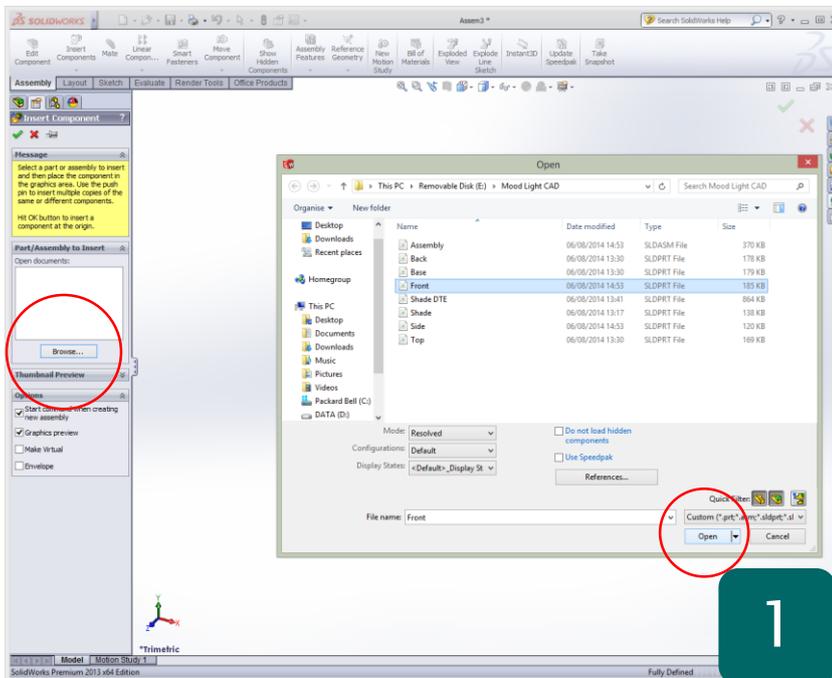
- File > New > Assembly. In assembly mode Solidworks allows for any number of Part files to be joined together with constraints
- Click '**Browse..**' in the left feature menu
- Access the parts folder from; Shared area > Design Technology> Mr Billington > Solidworks Tutorials > Mood Light
- Copy and Paste folder to your documents and open **'Front'**
- Open the part file and hold down the middle mouse button to orientate it in 3D
- Equally holding the left mouse button will allow you to select a view in 2D or 3D

Why? – By creating a set of Part files the model can have far greater complexity and realism when assembled and gives the user greater control when modifying their design

1
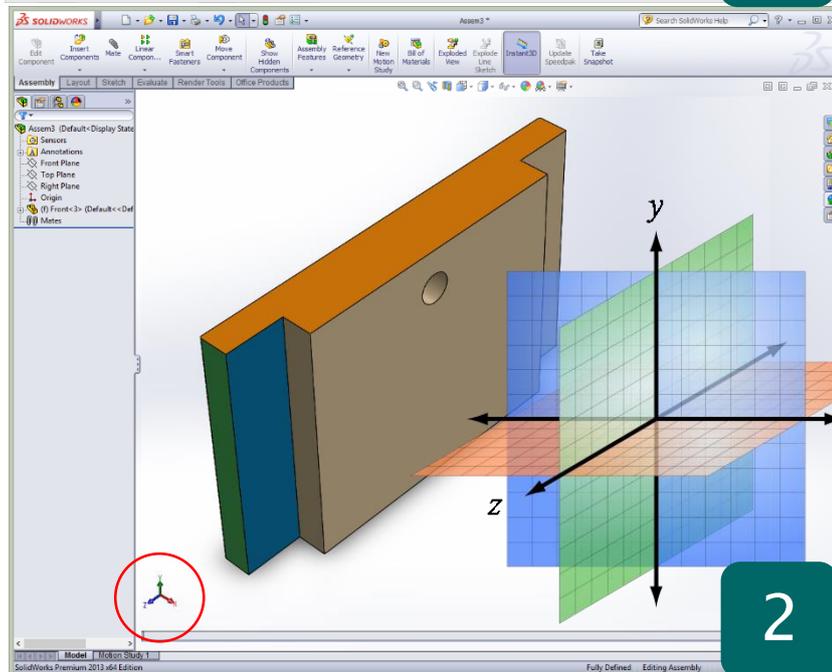
- Solidworks models in 3 dimensions; **X, Y & Z.** These are shown by the bottom left coordinate arrows
- In Assembly a part must be 'fully constrained' to prevent the parts moving unless this is required
- A position for each co-ordinate; **X, Y and Z** must be set for each part file. These are known as 'constraints'
- If a moving part is required only 2 constraints are needed

Why? – When a part file is imported it is suspended in 3D space free to move around. To assemble parts together they must be linked to each other as in the real world where glues and fastening are used.

2
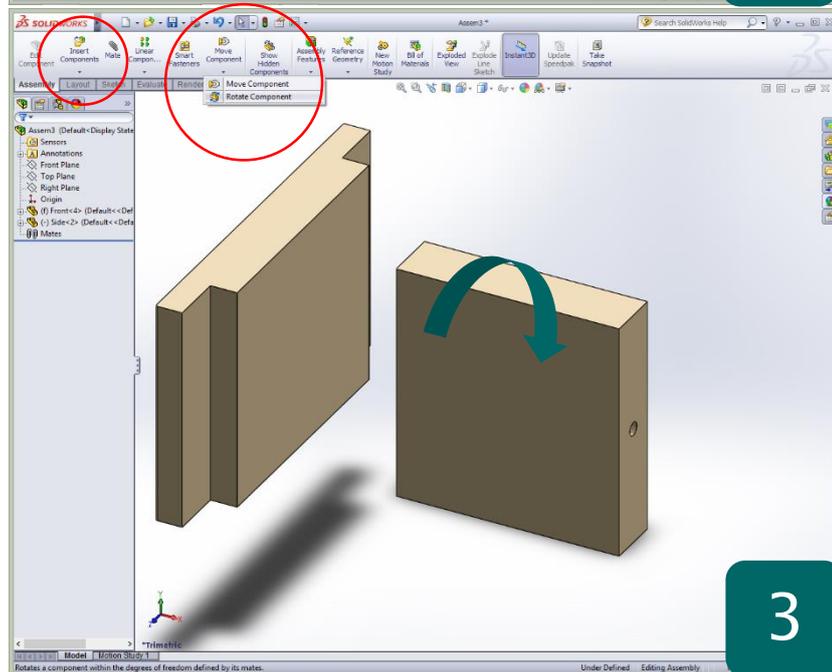
- Access the '**Assembly'** toolbar on the top left of the screen and click **'Insert Components'**
- Find the part file '**Side**' in the same folder and click open
- Confirm with the green tick
- Access the '**Assembly'** toolbar and click on the arrow below 'Move component'
- Select '**Rotate Component**'
- Use it to position the small hole in the side facing downwards
- Use the **'Move component'** feature to line up the side roughly with the lap joint

Why? – When assembling part files it is good practice to line up the part roughly where it needs to go before moving it into place. This avoids errors.

3
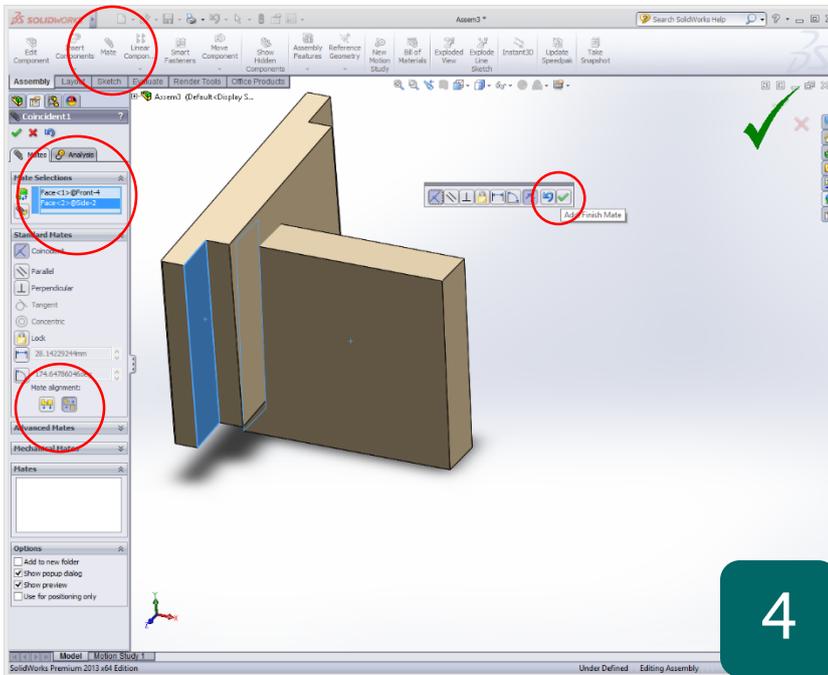
- Access the 'Assembly' toolbar and click on 'Mate'
- Click on the inside larger edge of the lap joint so that it turns blue
- Notice it is added to the feature list on the left
- Select the end of the 'Side' piece so that it turns blue, click the green tick to confirm
- The two pieces are now 'mated' by those faces on the models
- This is the first constraint
- If it mates the part in the wrong direction click the button under 'Mate alignment'

Why? – When assembling parts together you use the 'Mate' command to set a part of each model to be effectively 'stuck' together in the assembly
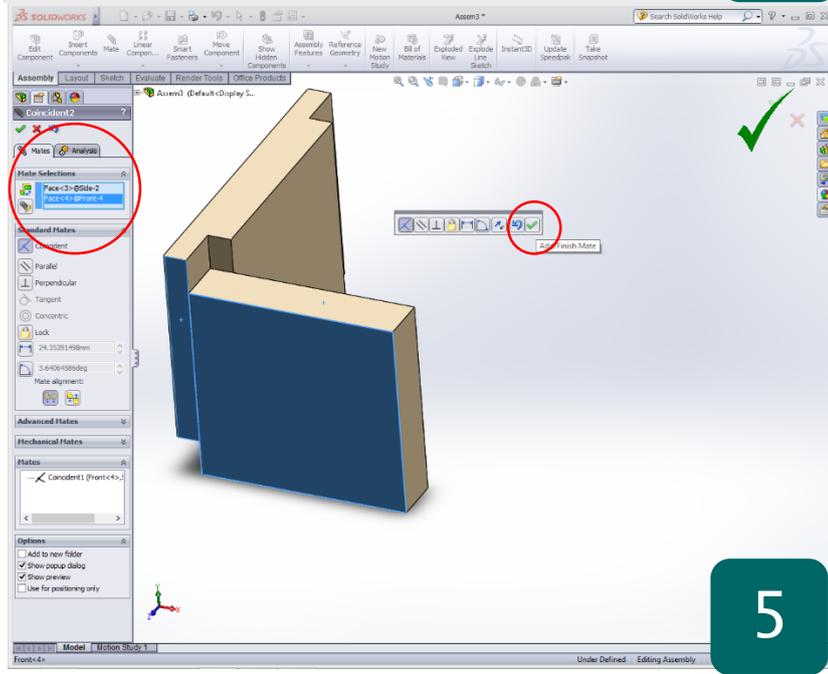

- Still in the 'Mate' tool now select the back of the 'Side' piece and the end of the 'Front' piece
- This is the second constraint
- Make sure you click the green tick to confirm
- Use the 'Move Component' and 'Rotate Component' commands to see how the part is now constrained
- Notice each time the 'Mate' is 'Coincident' in the left menu. This mean the parts meet at a point
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Why? - Mating the faces of each component together is the easiest and most robust way to ensure each is fully constrained
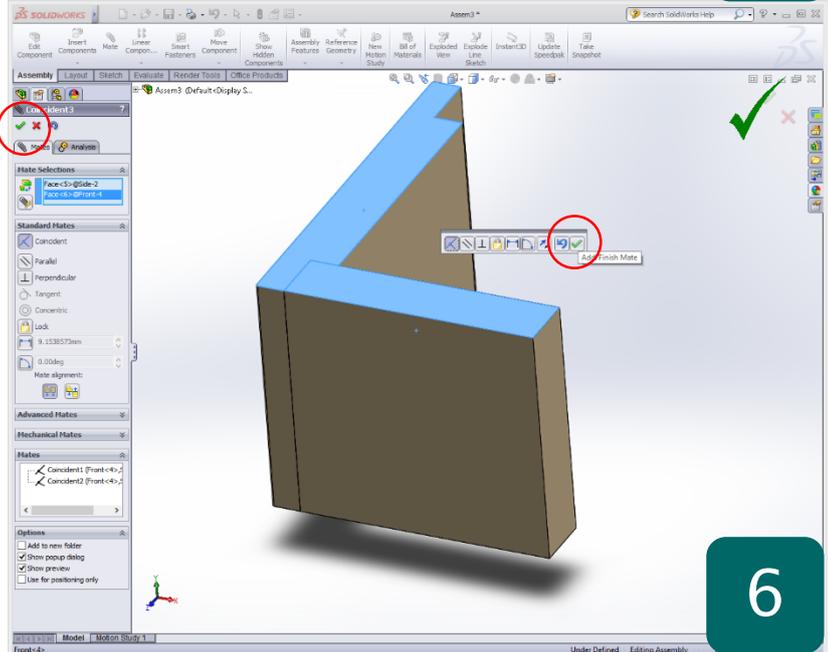
- Still in the 'Mate ' feature menu now select the two top faces to add the final constraint
- Confirm with the green tick
- Exit the feature with the green tick on the left menu
- Now use the 'Move component' and 'Rotate component command to confirm they are fully constrained
- They shouldn't move at all

Why? – If any part of an assembly is free to move in any direction it will be impossible to mate other components with it as neither will be fixed
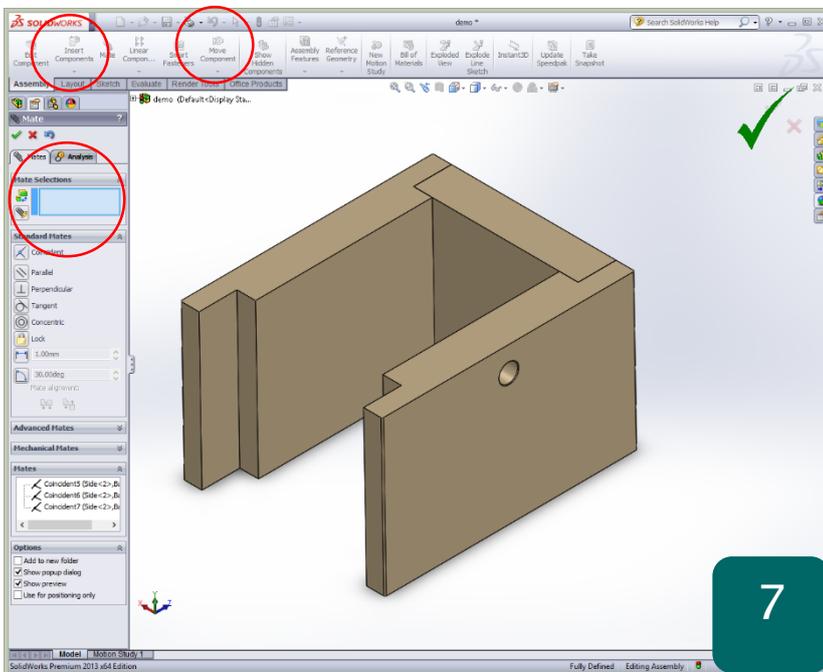
- Repeat the above **'mate'** process by adding the file '**Back**' from the part folder
- They should assemble as shown in the screenshot on the left
- You must assign **3 mates** to the part
- Rotate each part and move it roughly into place to ensure accuracy when assembling

Why? – Once you have got familiar with this process it should become very easy and repeatable across a range of different models. Use the middle mouse button to find the faces you need.
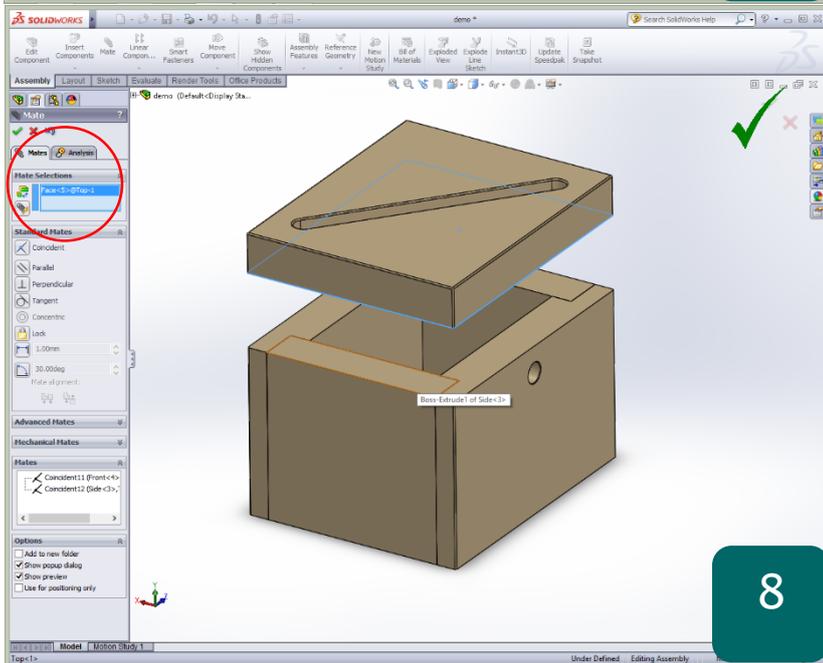
- Import '**Side**' again and **'mate'** to finish assembling the base structure
- Access 'Top' from the parts folder and use 'Rotate Component' and 'Move component' to get it in position
- Ensure the slot is facing up ^
- Use the 'mate' feature to fix the left edge
- Then the right edge
- Finally the base to the top of the structure
- Confirm each time with Green tick

Why? – Solidworks will automatically assign the correct constraint ie. Coincident depending on the parts of the model you are trying to Mate.
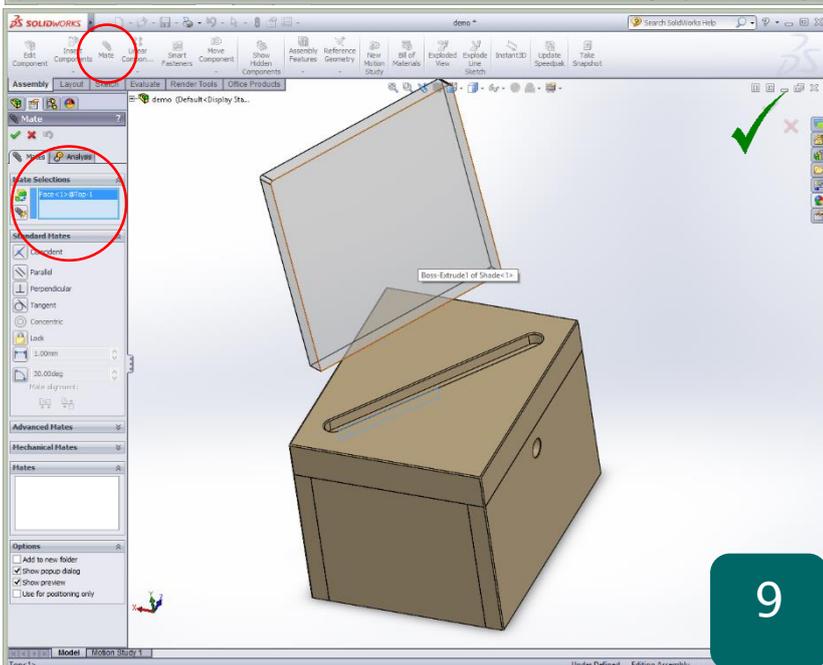
- Access the parts folder again using the 'Insert Components' feature
- Find the part file; 'Shade'
- Use the 'Move component' and 'Rotate component feature to line up the top as shown with the rounded corners at the top of the model
- Mate using the shade face > inside face of slot as shown
- Move to next step below

Why? – In order to mate the shade into the hole below so that it is fully constrained a different mate is required
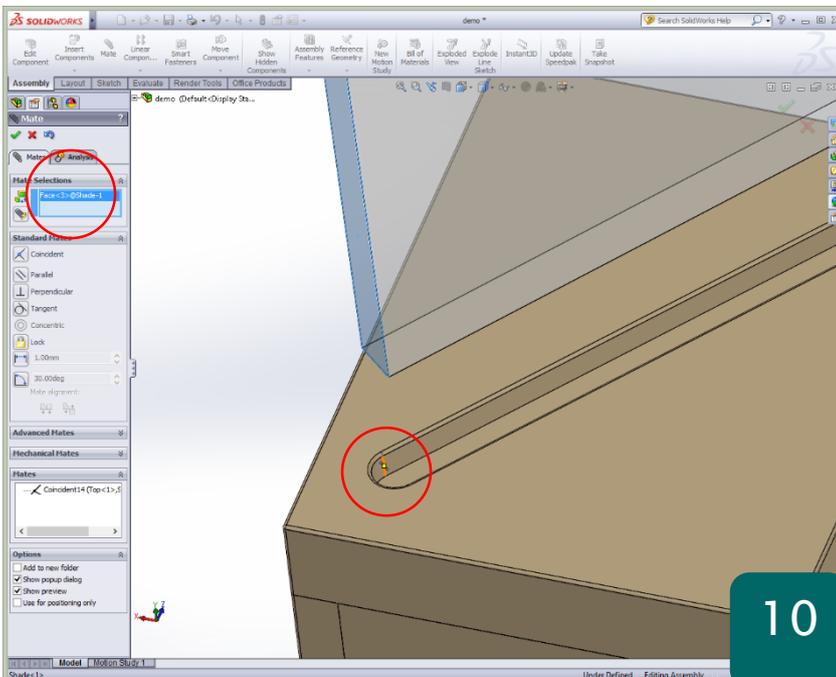
- Still in the '**mate**' feature select the inside vertical line shown one the left
- Mate this with the side of the shade part
- Confirm with the green tick
- Finally complete the Mate process using the bottom of the shade > base of slot

Why? – Solidworks allows you to mate contour lines as well as faces to fully constrain components together
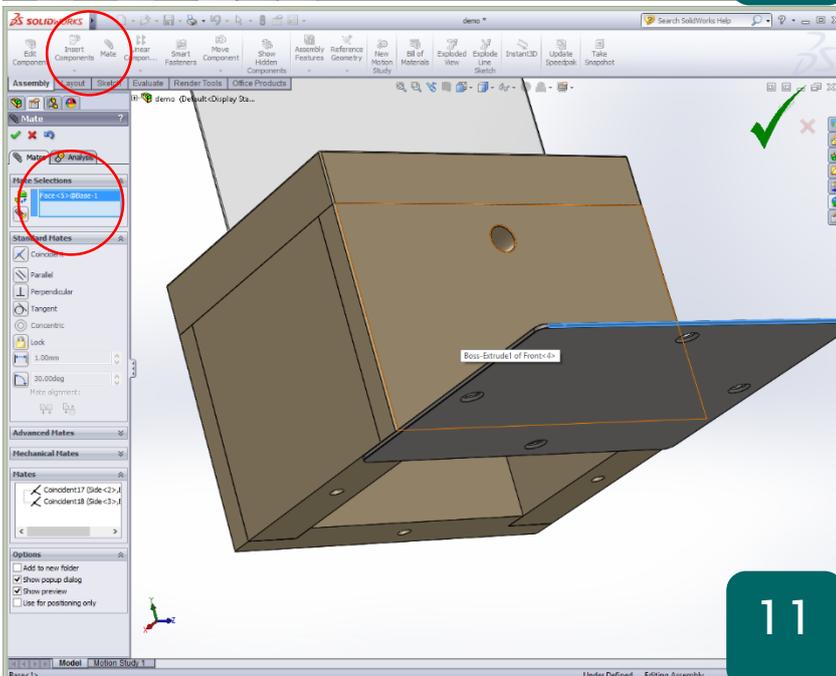
- '**Insert components**' from the 'assembly toolbar' and find; **Base**
- Rotate and position the base so that it lines up roughly with the bottom of the box
- Ensure the side with the 'chamfered' (sloped edges) holes is facing down
- Mate using; Long side > Long side
- Short side > short side
- Top > base of sides
- The holes should line up perfectly if done correctly

Why? - The chamfered side of the base is to allow the screw heads to sit flush with the base piece to prevent them scratching any surface.
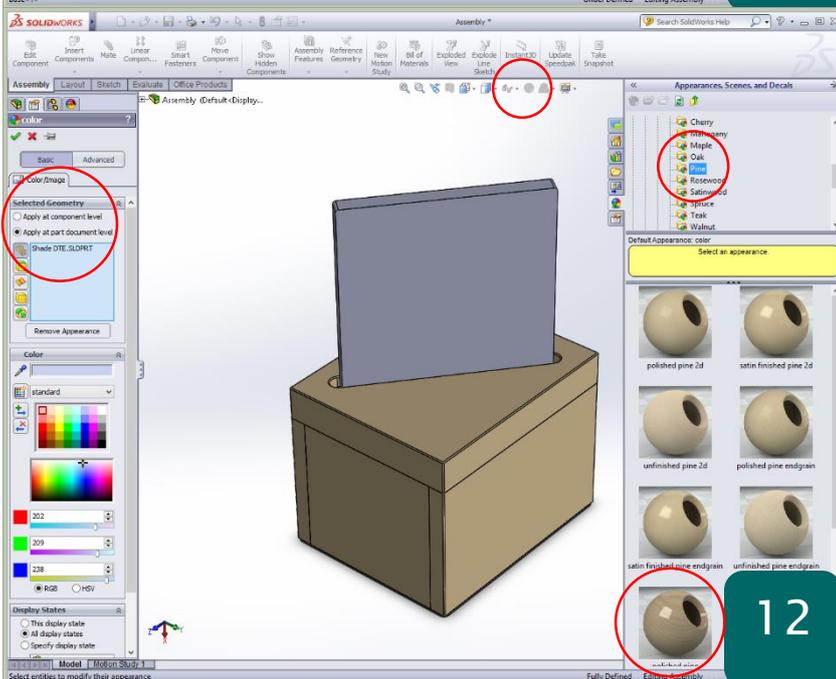
- The Mood light assembly is now complete and ready to be '**Rendered**'
- A render is when you apply colour and surface finishes to the model to make it realistic
- Click the coloured sphere in the middle of the screen
- On the right is the materials library, click the small **+** symbols to open each level
- Select '**Pine**' under '**Organic**' and from the bottom right previews find '**polished pine**'
- In the left feature menu select '**Apply at part level**'
- Select the part you want to apply the material to and double click on the material render ball to apply

Extension Task: Shade Design
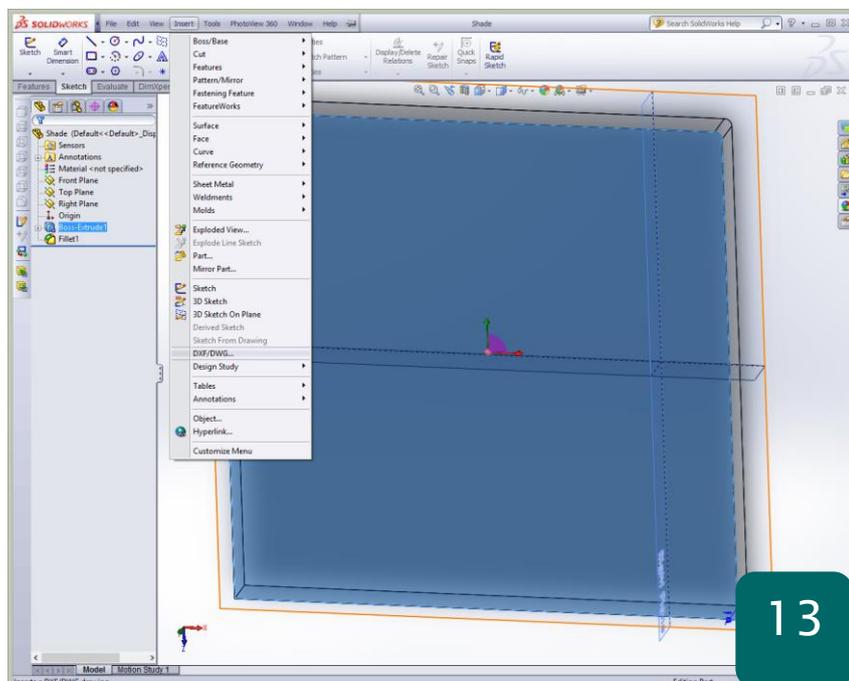
- *This step should only be attempted when you have completed the 'Dice' tutorial*
- To add a design to the '**shade**' you will need to open it as a part file
- In the left History '**Design Tree**' find '**Shade**', Click and a small toolbar should appear
- Click the first folder icon to '**Open Part**'
- Select the front square face and in the '**Sketch toolbar**' click '**Sketch**'
- Use the shortcut **CTRL + 8** on the keyboard to bring the view normal to.
- Click '**Insert**' on the top toolbar and scroll to **DXF/DWG**… and click

Why? – This part of the tutorial will show you how to add a 2D design sketch to you model
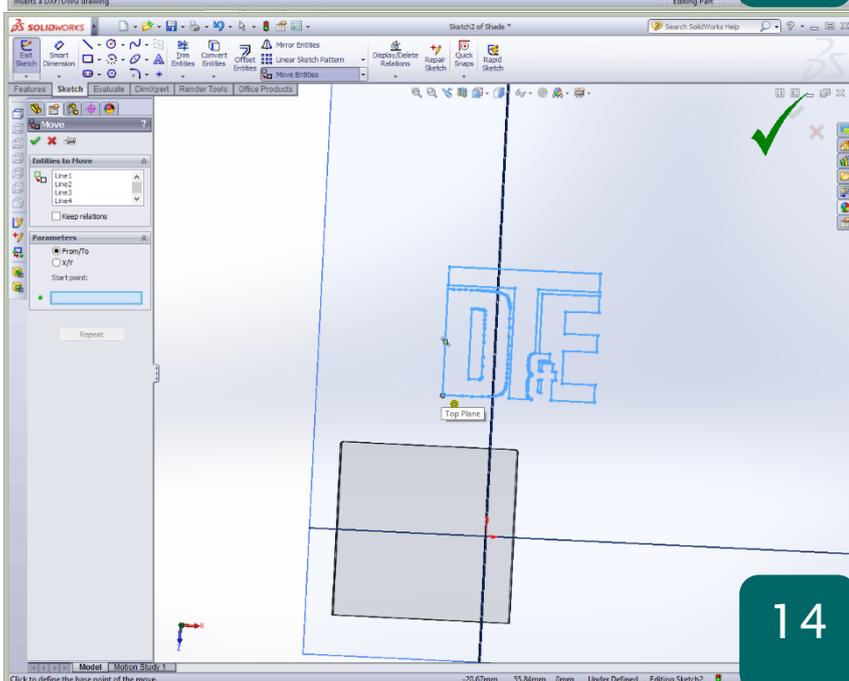
- Open the DXF file '**DTE**' in the file folder
- Click '**next**' in the popup menu
- Click '**next**' again and finally click '**Finish**'
- Once imported click and drag the selection box over the whole sketch
- Access the '**Sketch toolbar**' and click '**move entities**'
- Select the bottom left corner point as a '**start point**' and drag the sketch into position
- Click '**Exit Sketch**'

Why? – In 2D design it is easier to create complex geometry and sketch designs and save them as DXF files to import. Solidworks is better for 3D work
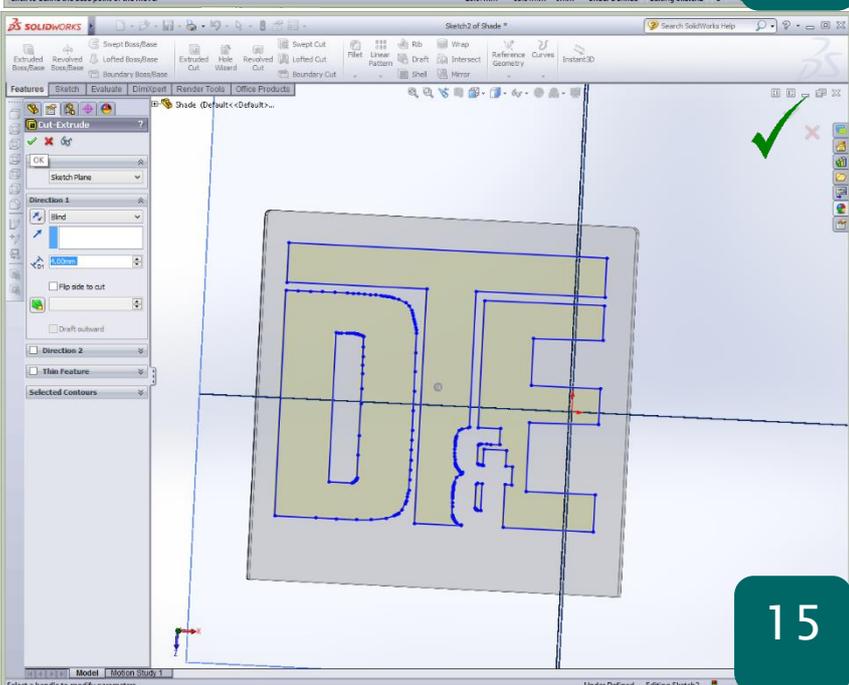
- Access the '**Features**' toolbar and click '**Extruded cut**'
- Set the depth to **4mm** to cut into the model
- **If** the features in going in the wrong direction click the double arrows in the left feature menu
- Confirm with the green tick
- Click '**Save**' on the part file and close
- Open the assembly completed earlier and the shade should have updated with the new design on top

Why? – The original assembly will constantly update itself depending on any changes you make to the part files associated with it. Be careful not to make any considerable changes to each parts shape as it will affect the 'mate' constraints you have used