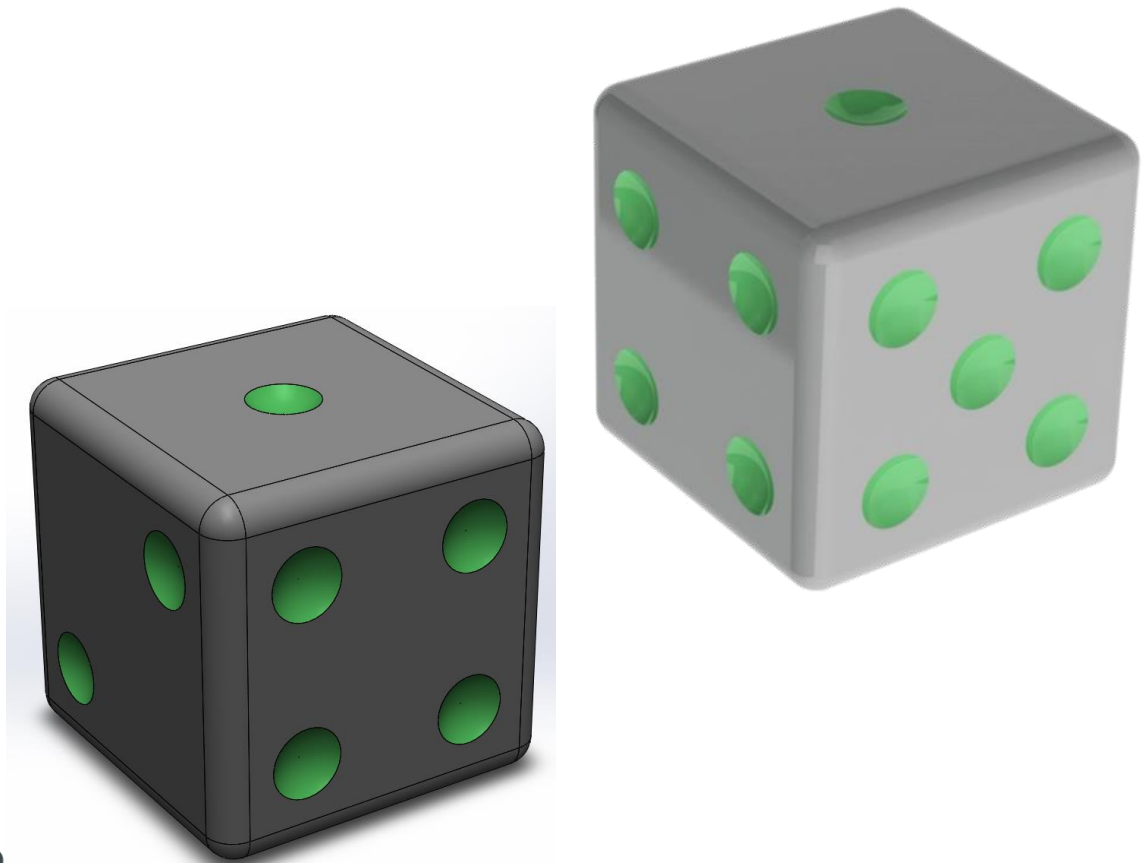


Solidworks/2014

3D Modelling Tutorial



SOLIDWORKS

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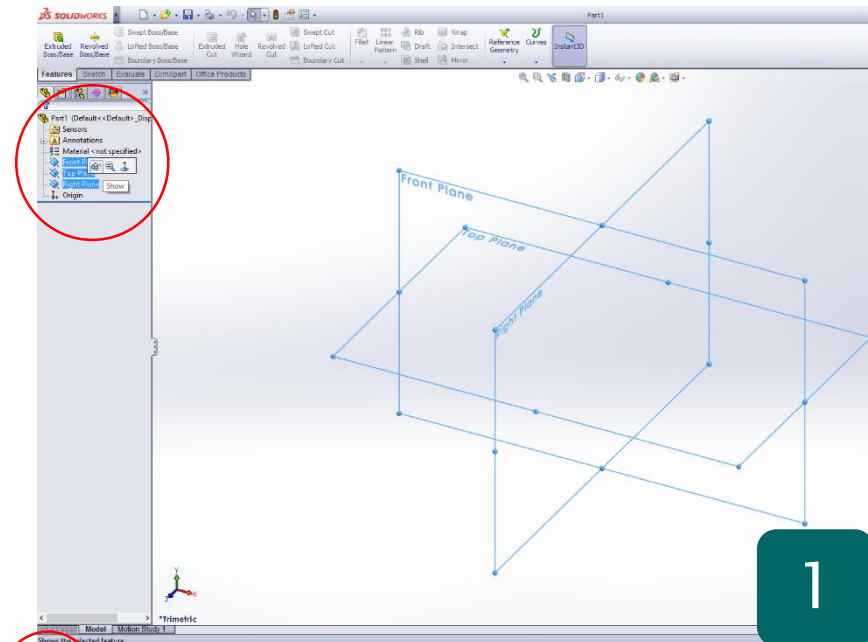
Learning Outcome; 6 Sided Dice

Skill Level; 1 - Beginner

3D; *Extrude Boss, Extruded Cut, Fillet (face/line), Render*

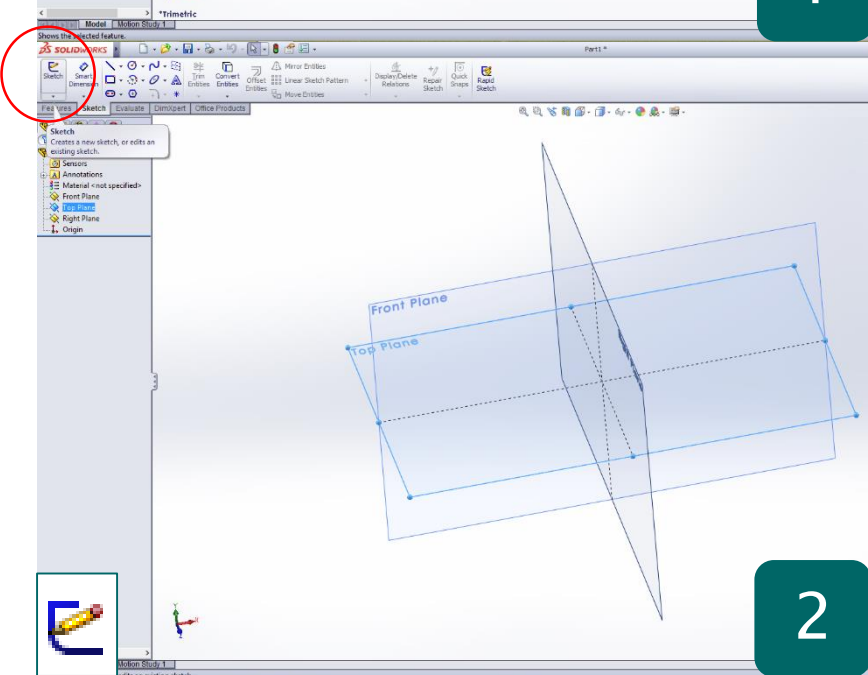
2D; *Planes, Smart Dimension, Line tool, Circle Tool, Mirror, Trim*





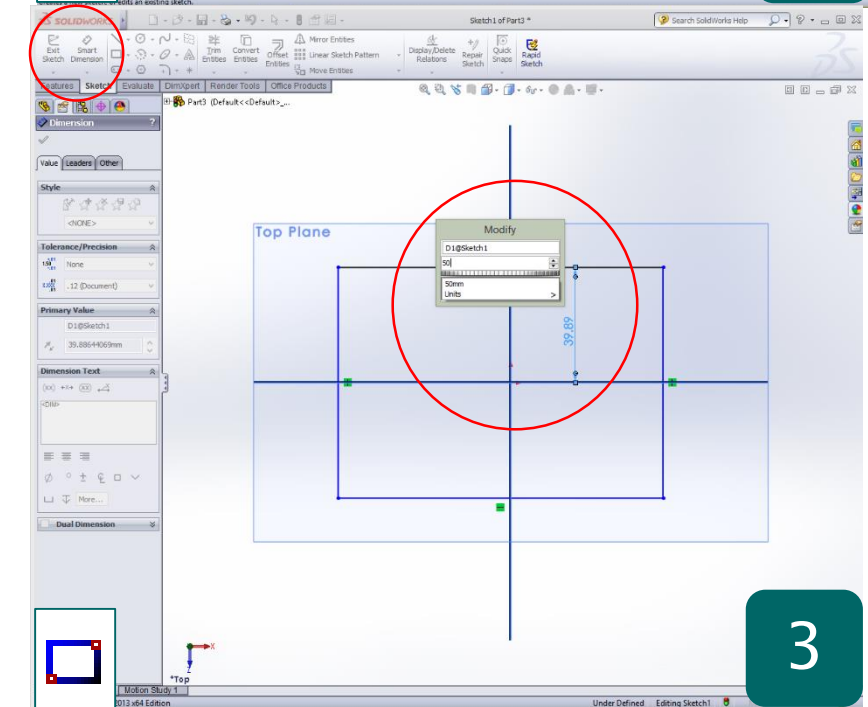
- Hold **SHIFT** on the keyboard and select all three work planes – **Front, Top, Back**.
- Hover over them in the History viewer and click the **'Show'** button shaped like glasses
- This will show all three planes in your workspace; X, Y & Z
- Practice using the scroll wheel in the middle of your mouse to click and rotate around your 3D world
- Holding the left button on the mouse will allow you to select a 2D view of your model

Why? – This enables you to clearly see the virtual constraints of your model much clearer and helps you to sketch the designs required for your model

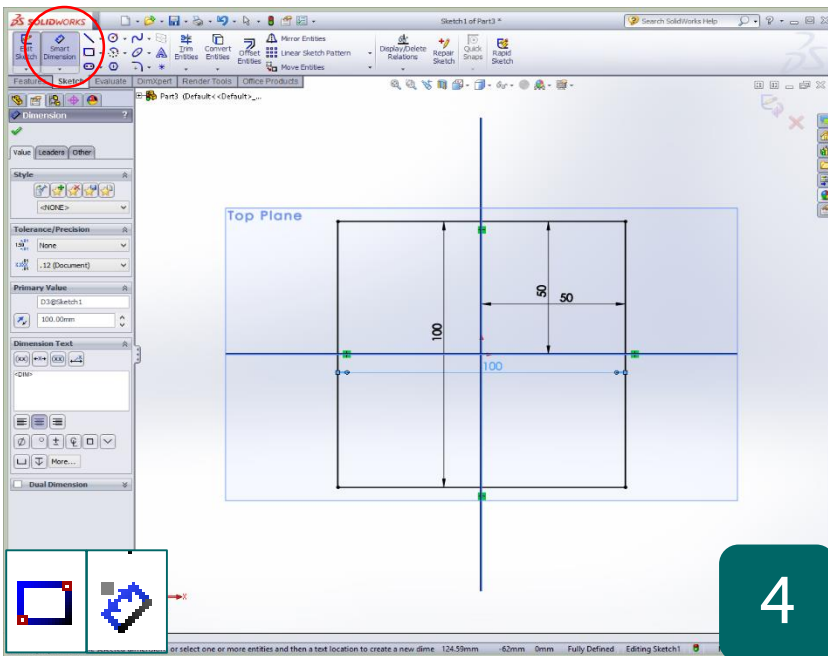


- Select the **'Top Plane'** in the work area and access the **'Sketch'** toolbar at the top left of the screen
- Click the **'Sketch'** button on the top left of the toolbar
- This will bring your workspace into a 2D view of the selected **'Top Plane'**
- Notice you will now be able to use the **'line tool'** to construct 2D sketches as well as the shape tools shown
- Test these new tools by randomly drawing on the workspace before moving to the next step

Why? – To create a 3D model you must first always sketch a 2D profile as a starting point. Sketches in 2D have no depth and therefore must form a complete (Closed loop) 'Profile' in order to form a 3D object



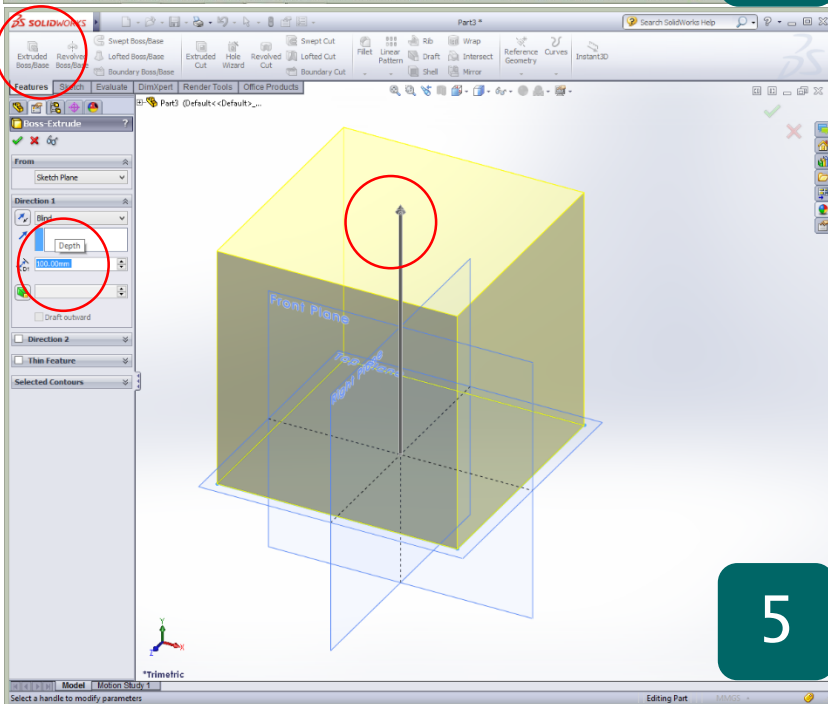
- Delete any practice sketches by click dragging across all and pressing the delete key on the keyboard
- Select the **'Rectangle tool'** and click draw a rectangle from the top left corner to the bottom right of the plane intersections
- To change its dimensions click the **'smart dimension'** button on the top left
- The tool allows you to select any 2 points and create a dimension between them which can be modified
- Click the top line of the rectangle and the centre plane line. A third click produced the dimension
- Type this as **50mm** and **Enter**.



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- Use the 'Smart Dimension tool' to further dimension the rectangle so that it measures 100 x 100mm
- Select the right side of the rectangle and the centre plane line with a third click to place.
- Type 50mm into the modify box and Enter
- Select the right hand side and the left hand side of the rectangle and dimension 100mm
- Repeat this with top and bottom to complete the 100 x 100 square for the dice.
- Click 'Exit Sketch' on top left of toolbar

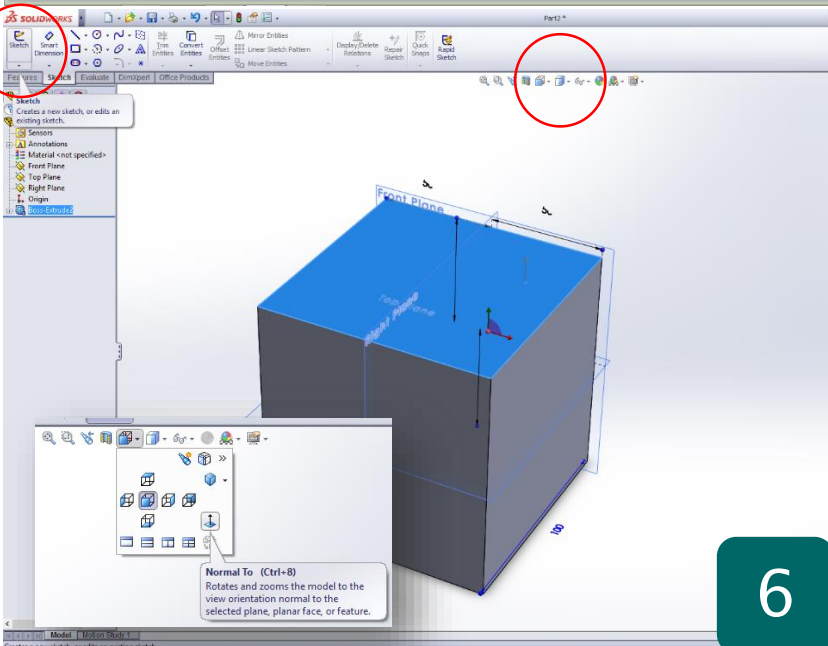
Why? – Although this may seem like a long process all dimensions should be related to your design planes. This make dimensioning later on in the model much easier and ensures a more robust model. *Alternatively you could double click any line to change its dimension*



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- Rotate around the newly created sketch and notice it has no depth
- Select the 'Features' menu of the top left and click 'Extruded Boss'.
- Select the newly created sketch and in the depth box on the left type 100mm
- Click the 'Tick' to confirm the feature
- You should now have a 3D cube in your workspace which you navigate around

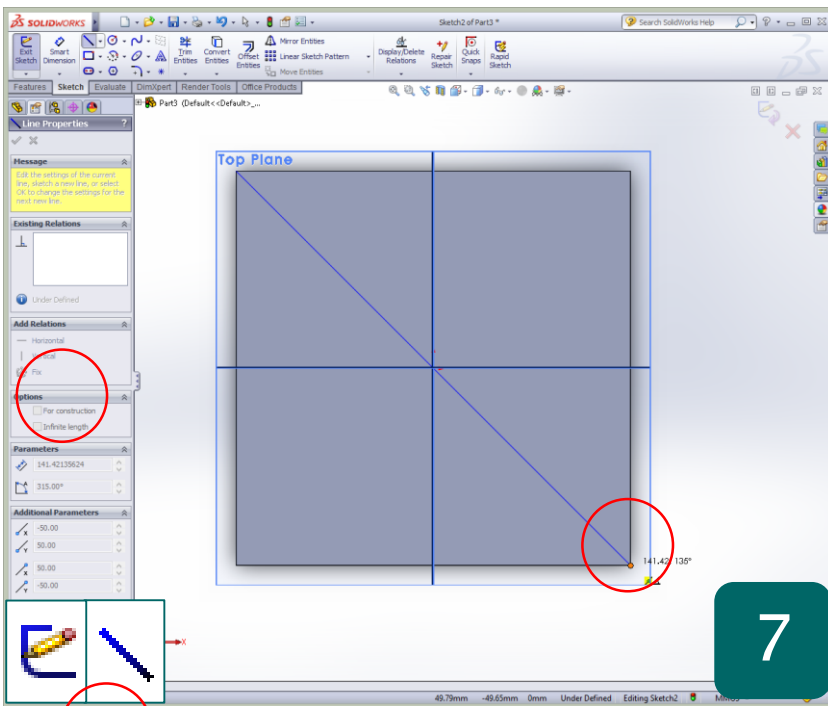
Why? – The Extruded boss feature works in a similar way to classic Playdoh toys. You set a profile (sketch) and you pull that in a set direction for as long as your design requires. Alternatively you can use the arrow shown in the picture to manually adjust the depth of your object.



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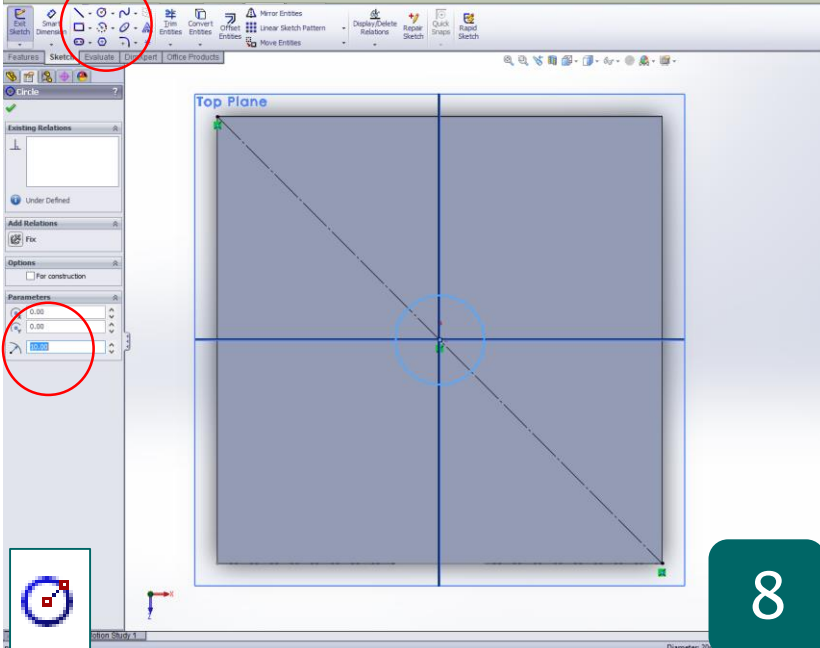
- Select any side of the 3D cube so that it turns blue
- Select the 'Sketch toolbar' and click the 'sketch' button on the far left
- Notice the workspace will not automatically switch to a 2D view as before
- Use the keyboard to orient the cube so that it is 'normal to' your sketch plane
- Hold 'Shift' and press '8' on the keyboard to orientate your view.
- This will bring you into a 2D view of the model face you want to sketch on

Why? – By selecting the face on the model first and then clicking sketch you can access any face on your model without creating a new plane every time. When sketching it is good practice to be 'normal to' the face ie. In a 2D view.



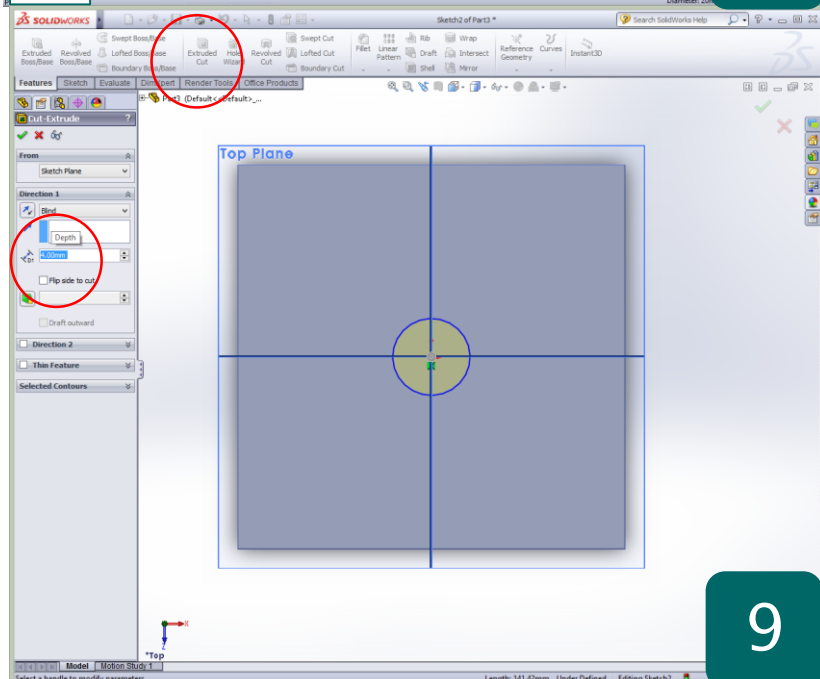
- Using the 'Line tool' hover over the top left corner of the square and notice an orange dot appears
- This will allow you to 'snap' to a point of the model such as a corner or end of line
- Using the orange dots draw a line from one corner to the bottom right diagonally
- Press the **ESC** key on the keyboard to end the 'line tool'
- This will for the construction line for the first number on the dice; 1
- Click back on the line so that it becomes blue and on the left hand side menu tick 'For Construction'

Why? – Solidworks allows you to use other dimensions in the model to quickly snap to when sketching. You can use this to quickly dimension and draw construction lines on your model. Selecting 'for construction' excludes it from the feature used.



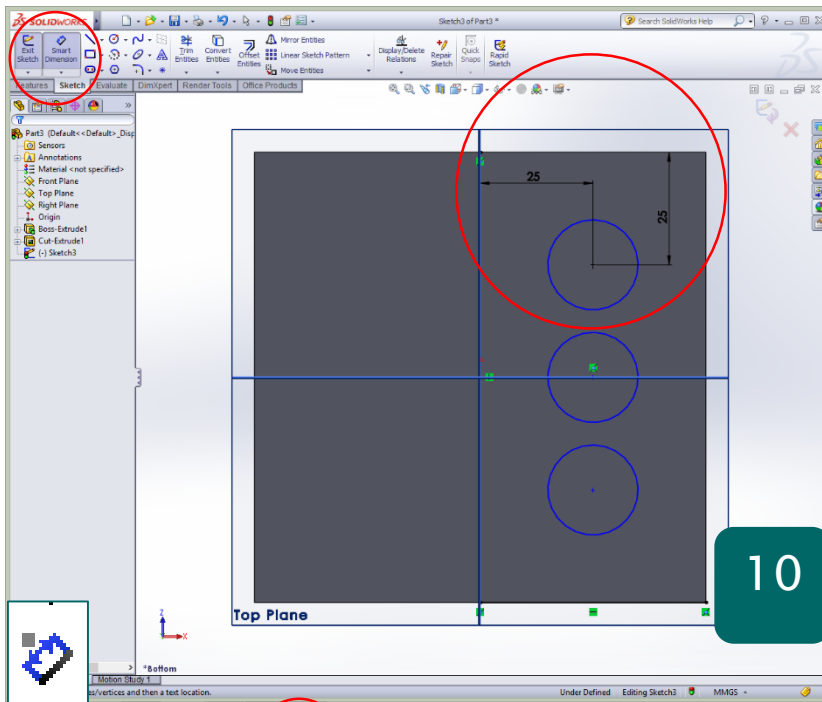
- Select the 'circle tool' in the 'sketch toolbar' which requires you to pick a centre for the circle
- Hover over the middle of the diagonal line and notice a yellow square appears – this is the 'midpoint'
- Click to place the circle and type **10mm** into the radius box on the left menu
- Click the green tick on the left to confirm this action

Why? – Mid-points are simply the middle of any line or shape and are incredibly useful when drawing lines for construction and accurate sketches.



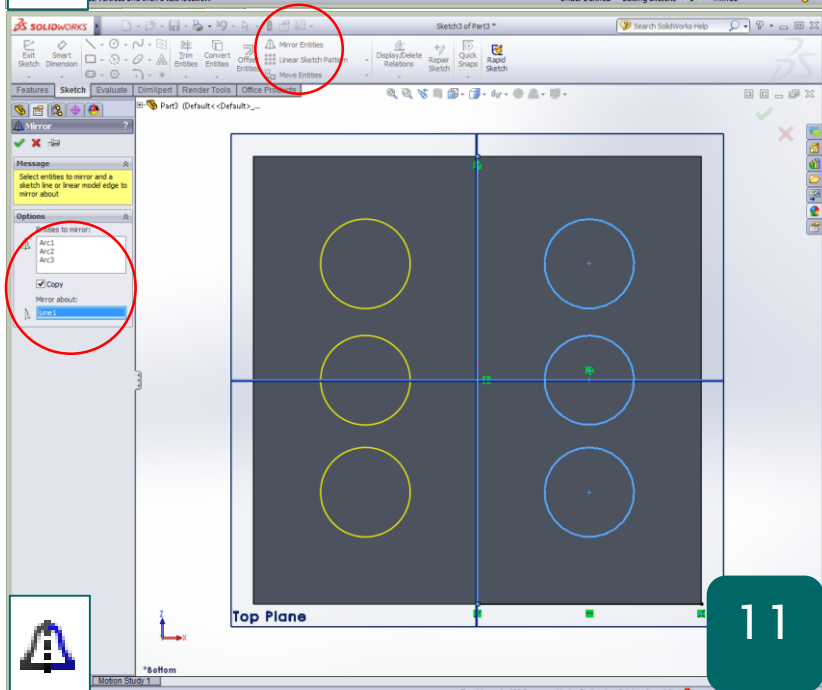
- With the circle selected access the 'Features' tool bar at the top left of the screen
- Click 'Extruded Cut' which should turn the circle yellow and suggest a depth on the left
- Type the depth as **4mm** into the left box and hit **Enter**
- Click the green tick to confirm this feature
- Use the middle mouse button to orientate your model and you should see the hole circle has created a hole in your cube

Why? – Extruded Cut does the opposite to Extruded Boss by removing material from a solid 3D shape. The direction it removes the materials can be altered by clicking the double arrow on the left and the depth can be adjusted manually using the arrow.



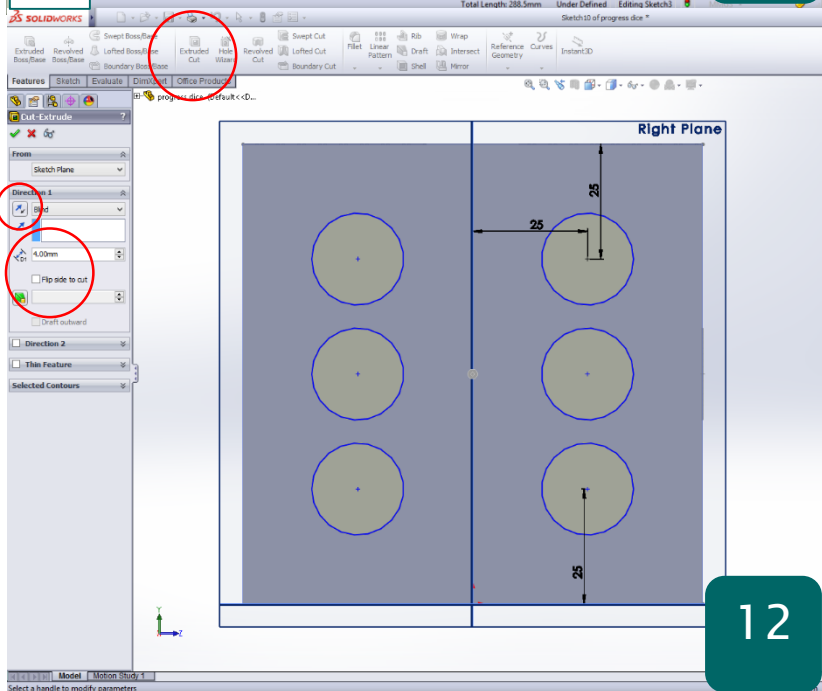
- All opposite sides of a dice add up to 7 and therefore 6 is going to be sketched on the opposite side to the 1 just completed
- Select this face as before and click **'Sketch'** remembering to use **CTRL + 8** to Normalize
- This time each corner circle is 25mm from the centre plane and edge as shown.
- Draw a circle near the correct location with a radius of **10mm** again.
- Use the **'Smart Dimension'** tool to set the middle of the circle 25mm from both the edge and the centre plane
- The middle circle snaps to the middle plane or if not visible draw a centre line using the midpoint of the side edges.

Why? – Each circle must evenly space on the cube to ensure an accurate model is created. Only 3 circles are required to move to the next step.



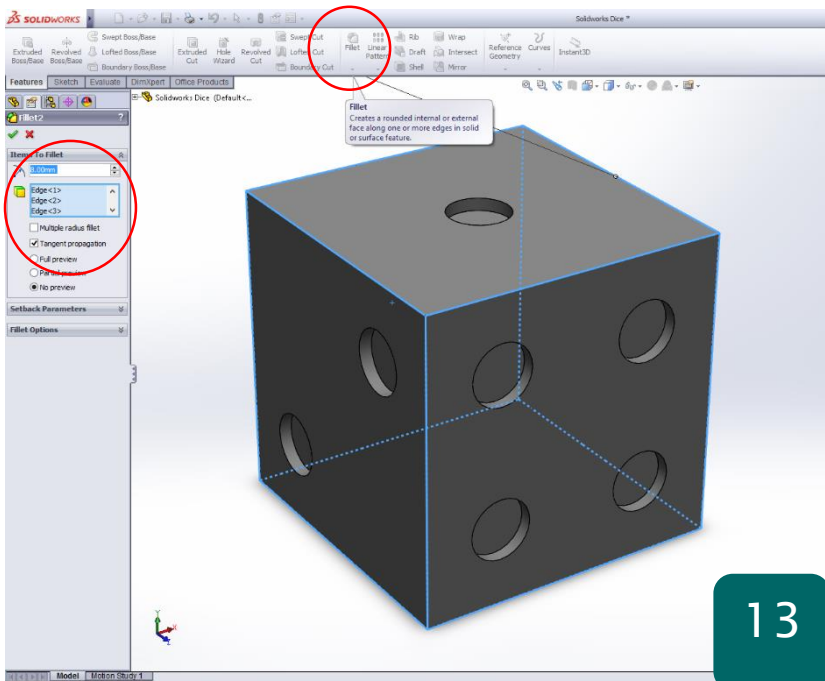
- Once all three have been correctly dimensioned they will now need to be copied using the **'Mirror Entities'** tool.
- If you do not have a visible plane going vertically down the middle of your cube draw a line again using the midpoints
- Click and drag the mouse creating a selection box over the three circles
- Click the **'Mirror Entities'** button on the top right of the sketch toolbar.
- Select the **'mirror about'** option on the left menu and select the middle vertical plane or your drawn line to reflect along
- If the preview matches the view on the left click the green tick to confirm

Why? – The mirror tool saves time when creating repeating geometry and is perfect for any symmetrical element of your design



- If you sketched a and extra lines in the last step select them and press the **DELETE** key on the keyboard to remove them
- Whilst still in the sketch select the **'Feature menu'** and as before **'Extruded Cut'**
- Set the depth to **4mm** and confirm with the green tick
- If no change has been made to your model repeat the action but change the direction using the double arrows on the left menu

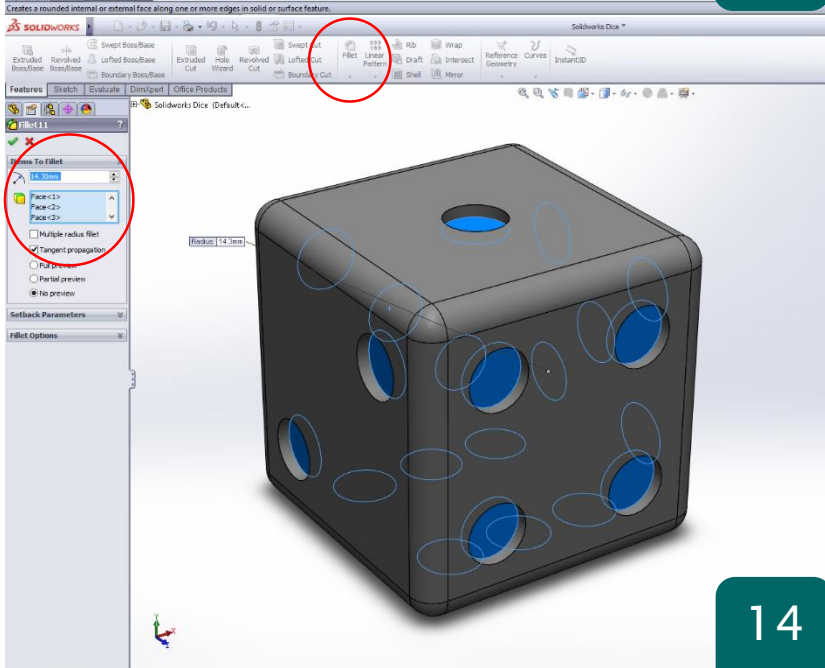
Why? – Sometimes Solidworks can assume the wrong direction for you feature to go. This can be easily flipped by clicking on the double arrow on the middle left of the features menu.



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- Repeat this process for the all other numbers on the dice ensuring all opposite sides add up to 7
- Once complete select the **'Feature Menu'** on the top left and find the **'Fillet'** button near the middle of the toolbar
- This will apply a round radius to any selected edges/faces on the model
- Move around the model selecting all of the outside edges of the cube so that they become light blue and add to the left list
- If you make a mistake just click it again to remove from the list
- When all are selected type; **'8mm'** into the radius box on the left
- If the preview looks correct confirm with the green tick

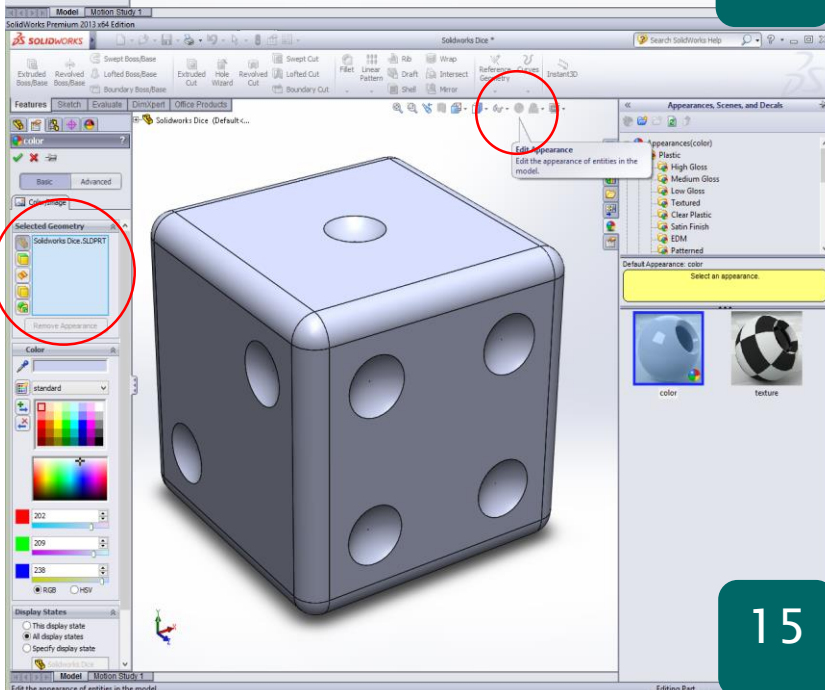
Why? – The fillet feature can make any model appear more realistic as every product edge will have a radius unless intended for cutting



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- Repeat this process but this time in the **'Fillet'** tool select the inner faces of the number holes all around the die
- Set the radius in the left feature menu to **14.3mm**
- Click the green tick to confirm the feature
- If filleted correctly the holes should now become have dome shaped with no clear face at the bottom

Why? – This last step could be done in many ways depending on the design of the die. Most real world dice have a concave hole for the numbers however this can be changed if required.



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- Now the model is complete you can apply the correct materials and colours you would like it made from
- This process is called **'Rendering'** and is started by clicking the **'edit appearance'** button (multi-coloured sphere)
- A library of materials will open up on the left and by clicking the little **+** symbols you can expand and explore into
- To apply a material to the whole part simply double click on the preview with the mouse
- To apply to a face on the part select **'Select Faces'** on the left menu and clear the part file from the menu
-

Why? – Applying materials to a file through rendering enhances the realism of your model.