Solidworks/2014
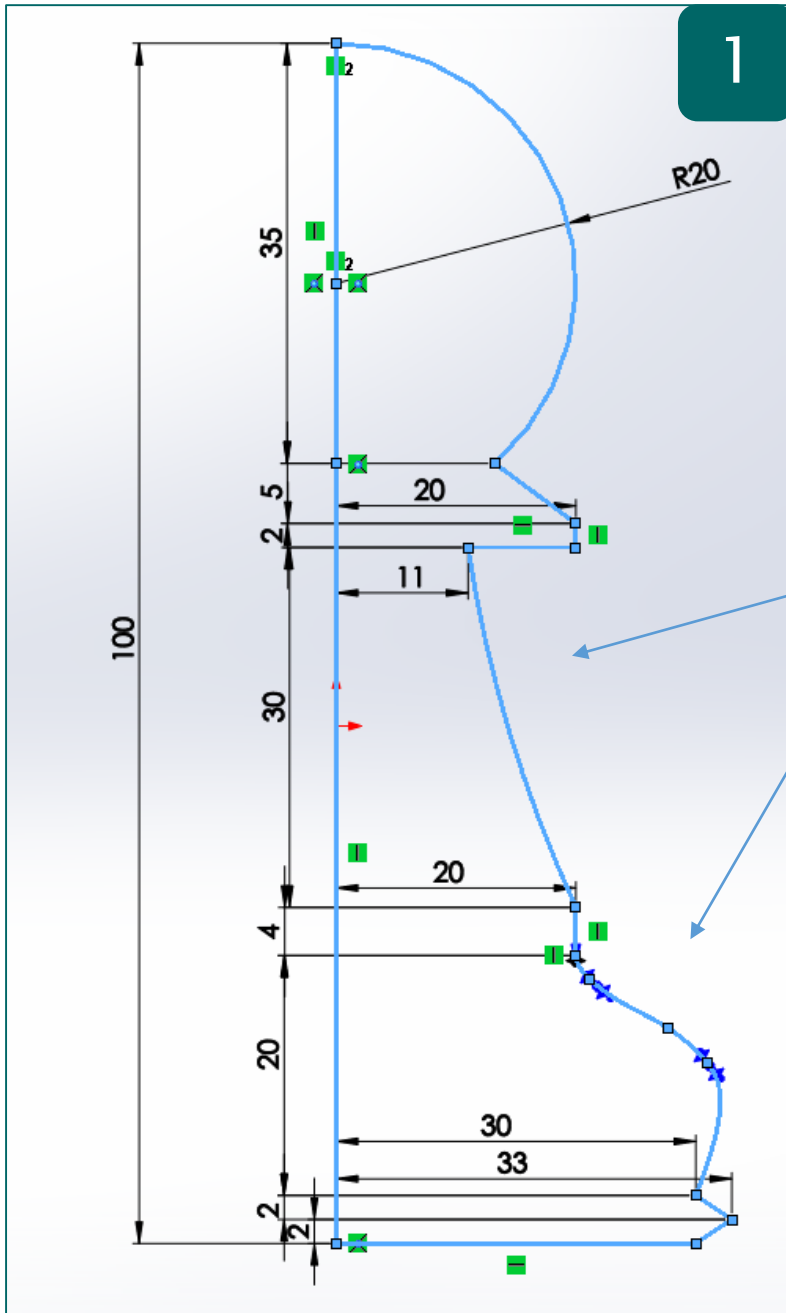
3D Modelling Tutorial

L2
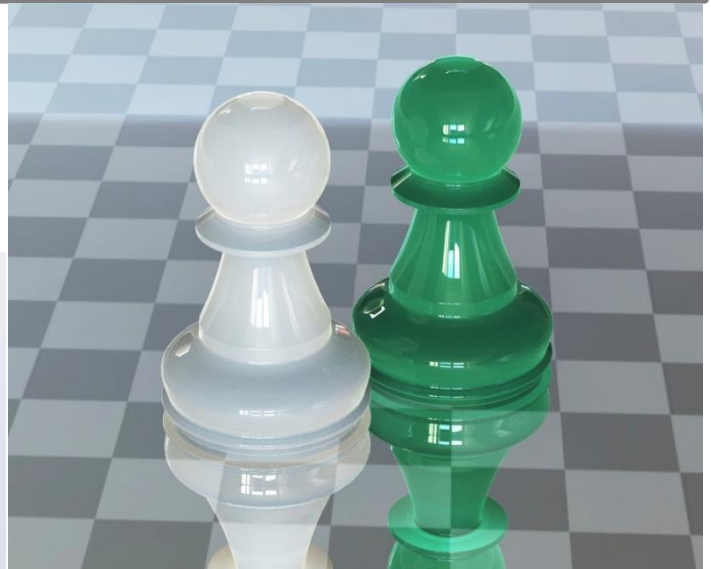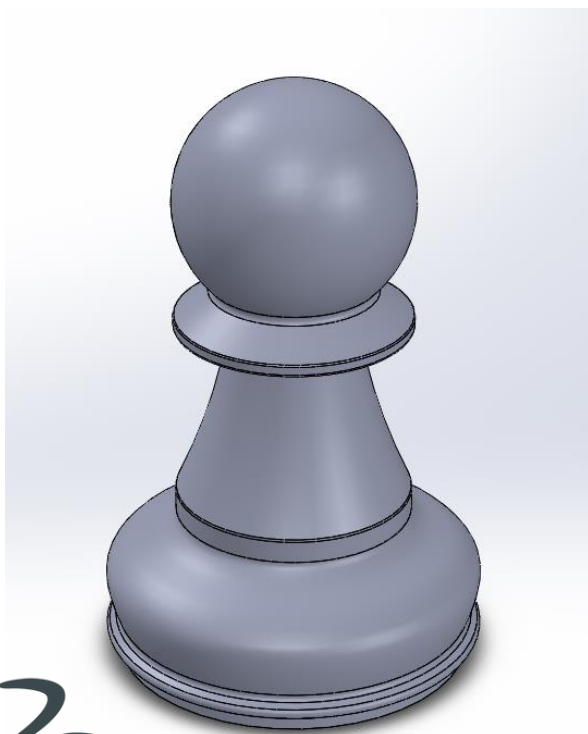
Learning Outcome; Chess Pawn

Skill Level; 2 Intermediate

3D; *Revolve Boss, Fillet, Render, Extruded Boss*

**2D;** *Sketch, Circle, Line, Power Trim, Dimensioning, 3 Point Curve, Spline*

DT&E

1

- This tutorial demonstrates how to effectively use the **'Revolve Boss'** feature to create complex geometry
- It also requires you to use a dimensioned engineering drawing to sketch from
- In a new Part select any **'plane'** in the workspace and select **'Sketch'** from the **'sketch toolbar'**
- Begin the sketch with a **100mm** vertical line snapped to the middle **'plane'** on your workspace
- From there use the **circle tool**, **line tool** and **Modify Dimensions** to match the drawing on the left

- *The **curve** in the middle is a 3 point curve explained below*
- *The **'spline'** at the base is also explained further in the tutorial*

Why? – The 'Revolved boss' features is a very easy way to create very complex circular geometry in a model. It requires a profile and a rotation axis to spin around. The rotation can also be controlled by changing the angle degree in the menu.

For it to work the sketch must be complete with no open ends or intersecting lines.
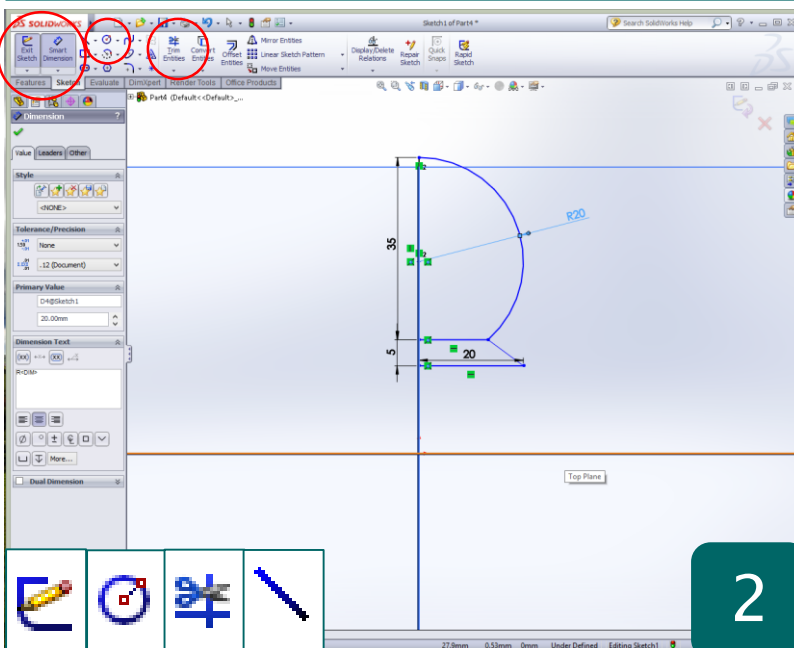

2

- Begin with a **100mm** line down the vertical intersecting plane
- Draw a **20mm** radius circle **20mm** down from the top
- Cut a line across at **35mm** from the top
- Access the '**Sketch**' toolbar and click '**Trim Entities'**. This will load a feature menu on the left
- Select **'Power Trim'** and click and move the mouse across the excess portion of the circle
- Sketch a line **5mm** down from the cut in and **20mm** long
- Join the two ends at an angle with a line

Why? – Power Trim is an effective way to quickly remove parts of the sketch that you no longer need. Use it like a pair of scissors and slide through lines to remove them

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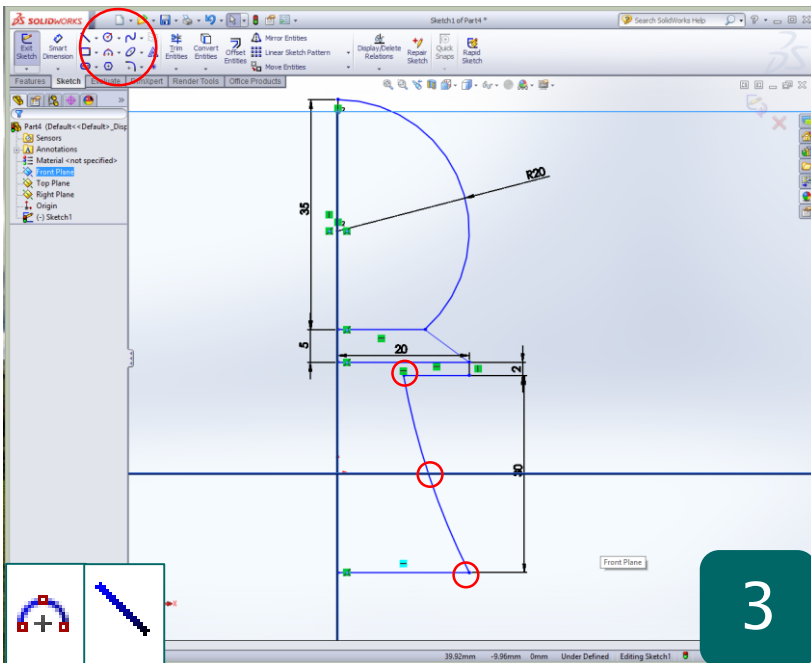
- Sketch a line **2mm** down from the angle
- Sketch another line **30mm** down and **20mm** long from this line
- Sketch a line **9mm** in towards the axis
- Access the **'sketch toolbar'** and click the arrow next to the '**centre point arc'** symbol
- In the drop down menu select **'3 Point Arc'**
- Click the first point as the end of the top line, Second point as end of bottom line
- Roughly match the curve to the one shown with the third point

Why? – The three point arc ensures a constant curve between two points irrelevant of the angle between them.
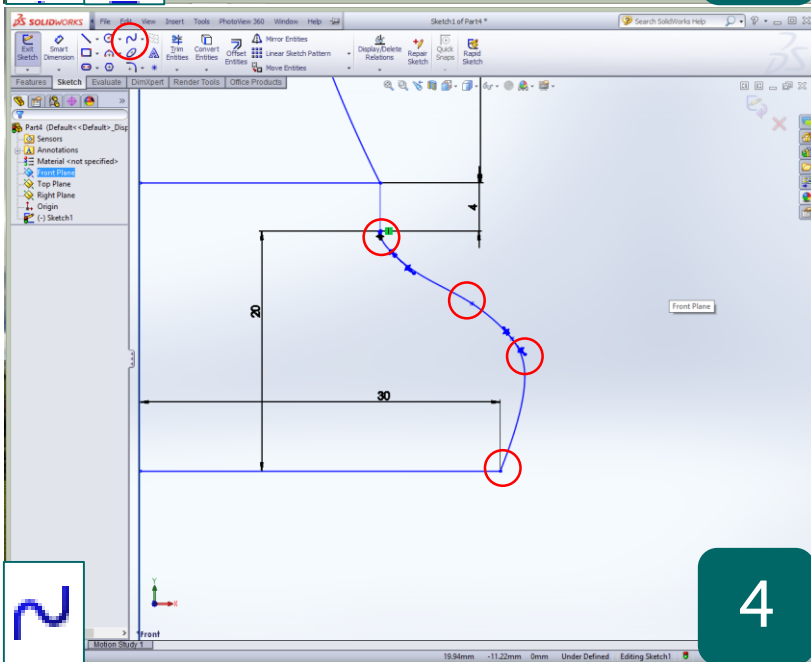

- Sketch a further line **4mm** down from the 3 point arc
- Access the **'Spline'** tool in the **'sketch toolbar'** and click to select
- To use the 'Spline' tool start from the end of the vertical line and place around 3 dots finishing at the bottom line
- It is impossible to dimension a spline so is needs to be rough guess
- Hit **ESC** on the keyboard to end

- Now you can edit each of the '**Spline'** points by clicking and moving them
- The arrows can be moved to change the tangency of the curves
- Click the top point arrow and in the left feature menu click '**Vertical'** under **'Add relation'**
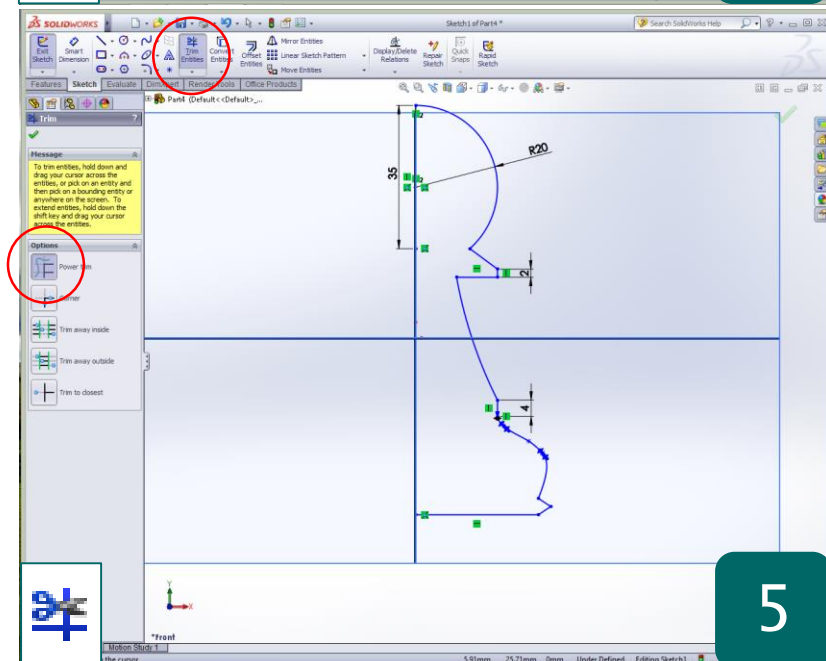
- Click on **'Trim Entities'** again and this time remove any extra parts of the sketch
- You should be left with just the sketch profile shown and have no open ends or intersecting lines
- Check that your centre line runs the length of the profile

Why? – When creating a sketch profile the most common mistake is open ended sketches and intersecting lines that have not been deleted

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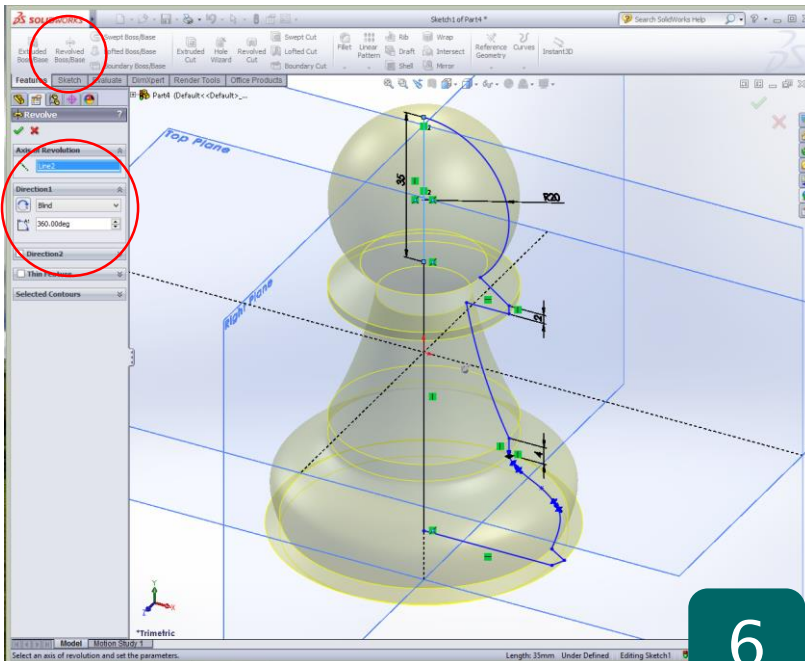
- Access the '**Feature toolbar**' and click on '**Revolved Boss**'
- Select the centre line as the '**Axis of Revolution**'
- Set the rotation angle to; **360°** if not already done so
- Confirm with the green tick if the preview matches the screenshot
- If the complete revolve looks different for any reason – click the '**+**' symbol next to the feature in the model tree and click the sketch
- In the menu that appear click '**edit sketch**'

Why? – A Revolved boss is a very simple and effective feature to use, but only if the sketch profile is accurate. If dimensions and lines are incorrect the shape may seem odd and distorted when revolved.


- Access the **Features toolbar** and select '**Fillet**'
- Apply a '**fillet**' to the contours shown in the selection box
- Set the radius to **0.8mm** to enhance the realism of the chess piece
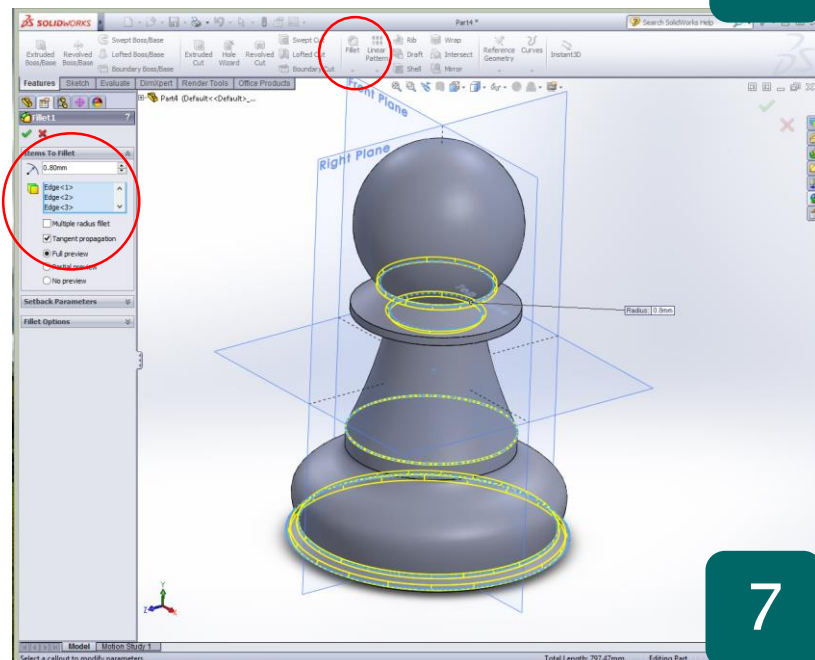
Why? – It is good practice to add a fillets to a model to enhance its overall realism as previously discussed. Even a slight radius such as the one above will make a considerable difference to the overall render of the model.

- Finally select the base of the pawn piece and click '**Sketch**' in the sketch toolbar
- Use the shortcut **CTRL + 8** to bring the workspace view normal to the screen
- Draw a circle from the centre of the model and snap onto the outer edge of the base
- In the '**features toolbar**' click '**Extruded Boss**' and set the depth to **2.00mm**
- This completes the model
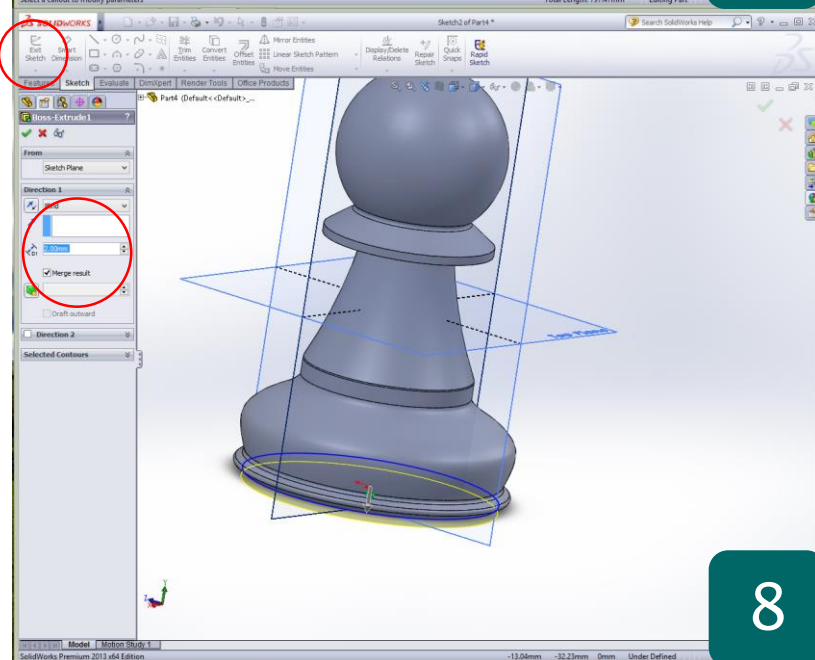
Why? – This step could have been added at the sketch profile stage with a simple line however it can be easily rendered separately as a base in the render process as a separate feature