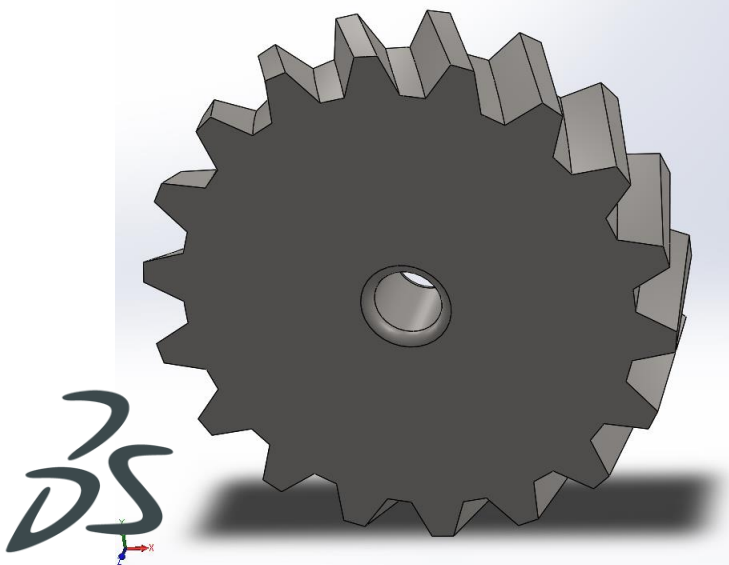
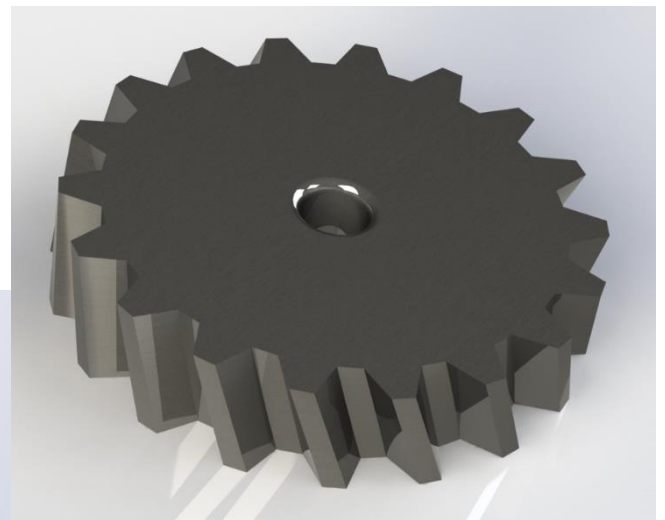


# Solidworks/2014

## 3D Modelling Tutorial

L2



**SOLIDWORKS**

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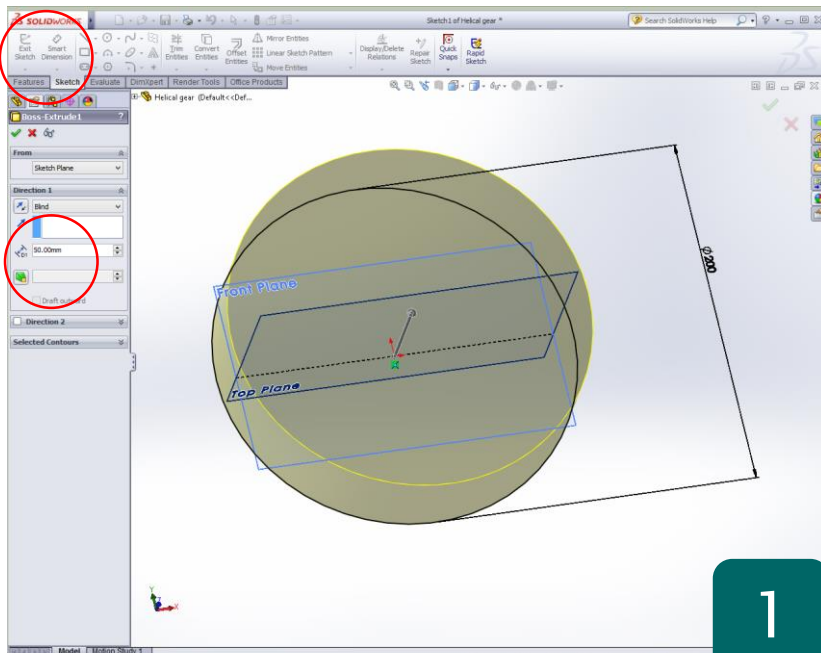
**Learning Outcome; Helical Gear**

**Skill Level; 2 - Intermediate**

**3D;** *Lofted Boss, Circular Pattern, Cut Extrude, Fillet*

**2D;** *Convert Entities, Rotate Entities, sketch, Smart Dimensions*

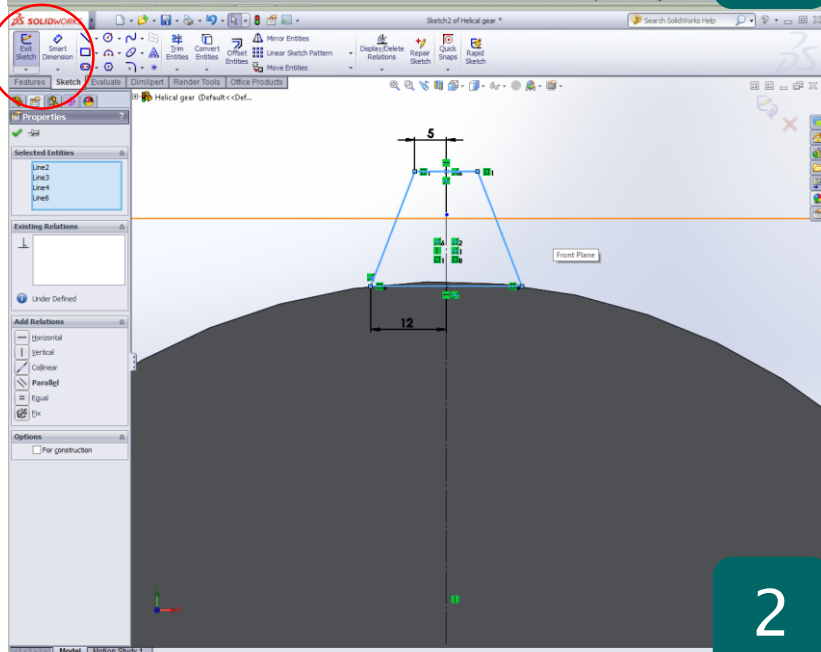




1

- Open a new **'part'** file
- Show all work planes in the model tree
- **'Sketch'** a circle with a **100mm** Radius
- Extruded boss to a depth of **50mm**
- Confirm with the green tick

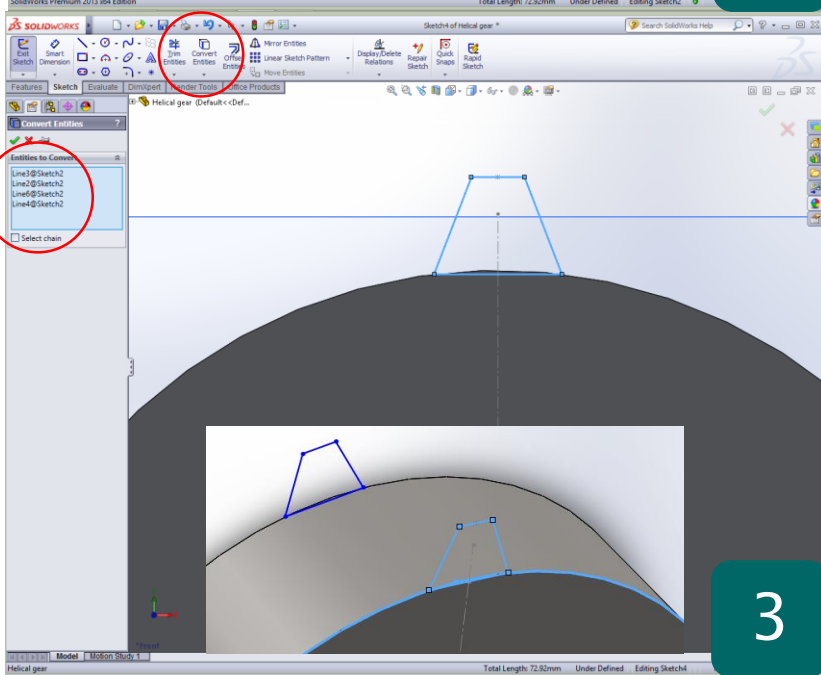
**Why?** – The feature in this tutorial; **'Lofted Boss'**, allows you to go from one sketch profile to another with different dimensions creating a 3D path in-between. The profiles can equally be different shapes creating complex and unique geometry



2

- Select the front face of the cylinder
- Click **'Sketch'** and use **CTRL + 8** to bring the workspace **'normal to'**
- Using the **'line tool'** sketch a line from the centre of the circle to directly up to the edge. Make **'for construction'**
- From the line sketch the left hand side of the shape shown and use **'Smart Dimension'** to make **5mm** and **12mm**
- Set the height to **18mm**
- Select all and use **'Mirror Entities'** to copy geometry using the construction line

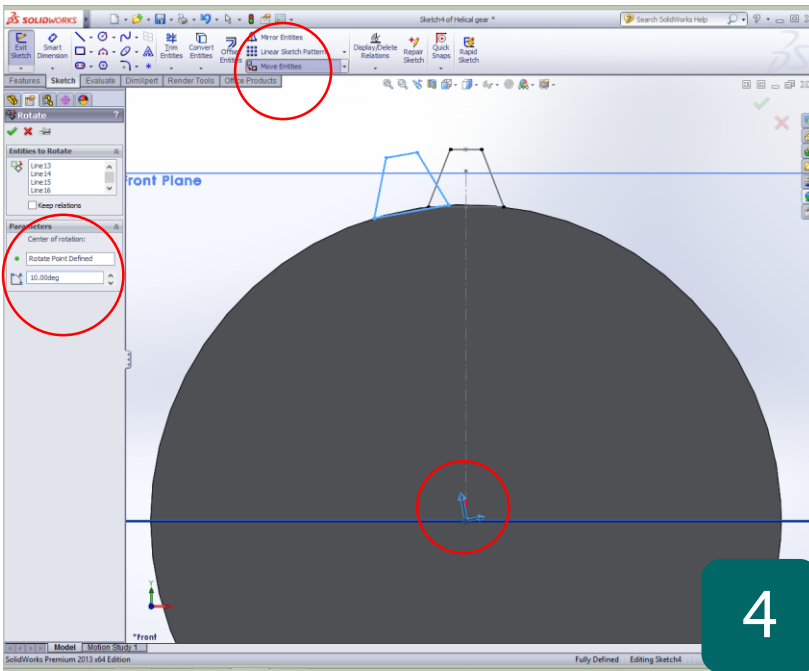
**Why?** – This will form the tooth profile for the gear and could be edited to suit whatever application. It must be mirrored to ensure accuracy and that it is tangent to the outside of the circle



3

- **Exit Sketch**
- Select the back face of the cylinder and click **'Sketch'** using **CTRL + 8** again
- In the **'sketch toolbar'** click **'Convert Entities'**
- Select each of the lines from the first sketch
- Confirm with the green tick

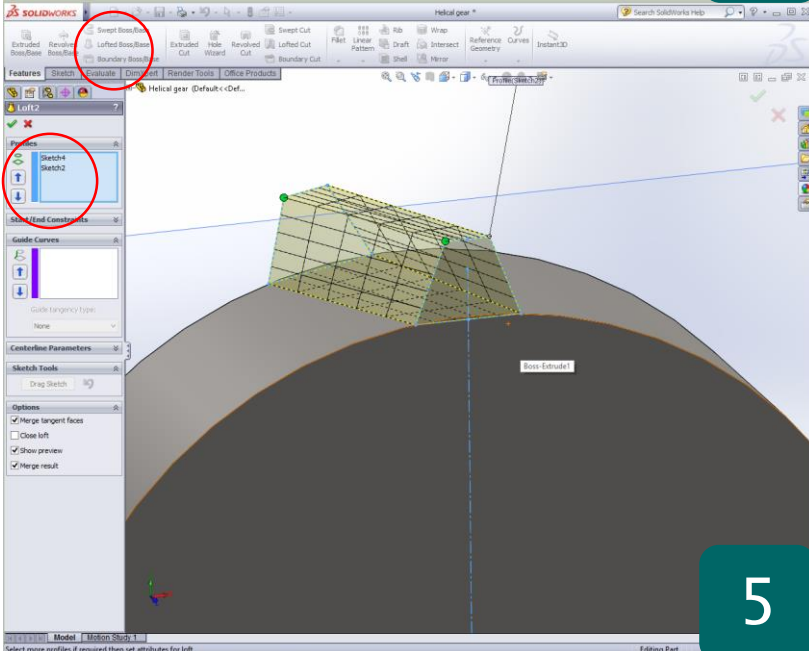
**Why?** – using the **'Convert Entities'** tool allows you to take geometry from any part of the model and turn it into sketch lines which can be modified. This is particularly useful when copying curved geometry.



4

- Select all the new lines and click the arrow next to **'Move Entities'**
- Select **'Rotate Entities'**
- Set the centre of the cylinder as the **'centre of rotation'**
- Set the angle of rotation to **10°**
- Confirm with the green tick

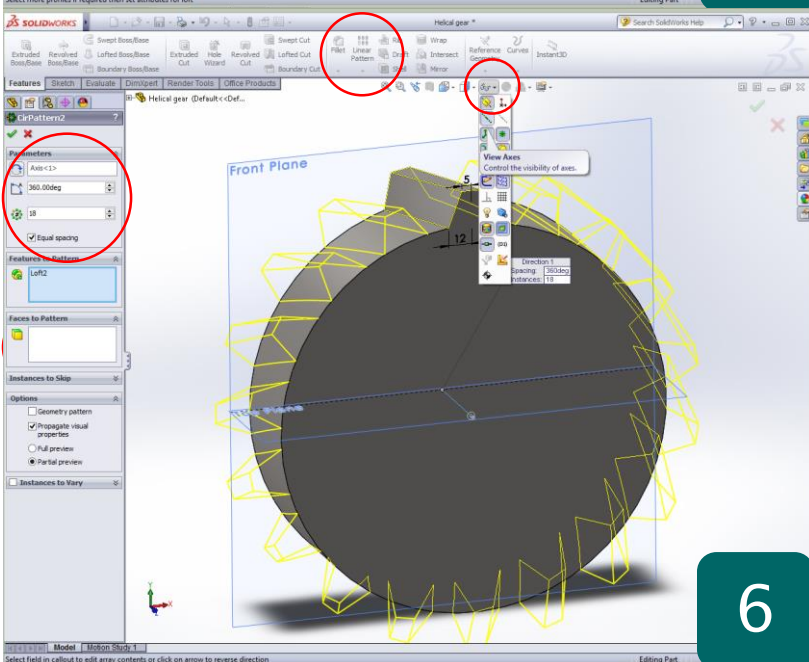
**Why?** – The rotation of the second sketch creates the helical profile of the gear and can be adjusted to suit. A simple extrude would create a straight toothed gear if required.



5

- Access the **'features toolbar'**
- Click **'Lofted boss'** to load the feature menu
- This allows you to create a 3D shape between two profiles without a path
- Select the two sketches so that they appear in the feature menu
- Confirm with the green tick if the profile matches that in the screenshot

**Why?** – This feature is useful when you know the start and end profile of shape as the software calculate the appropriate transistor between the shapes. A guide curve can be added to change its path



6

- In the **'model tree'** click on the **'lofted boss'** feature
- Access the **'features toolbar'** and click on the arrow under **'Linear pattern'**
- Select **'circular pattern'**
- Now to find the centre axis of the cylinder turn on **'axis'** in the **'hide/show'** menu
- Select the axis as the **'rotation parameter'**
- Tick **'space evenly'**
- Type **18** into the number of entities box
- Confirm with the green tick

**Why?** – The pattern tool can be used in many ways to quickly duplicate a feature on a model. The final features are optional and can be applied for the render by adding a mounting hole and material