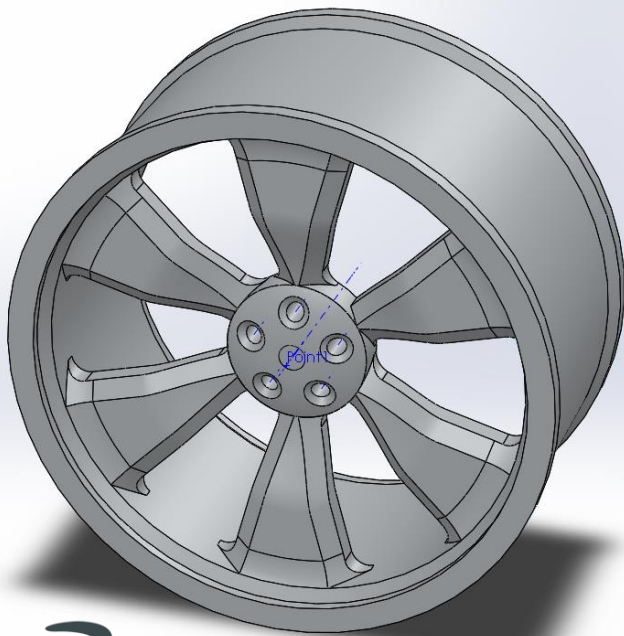


Solidworks/2014

3D Modelling Tutorial

L2



SOLIDWORKS

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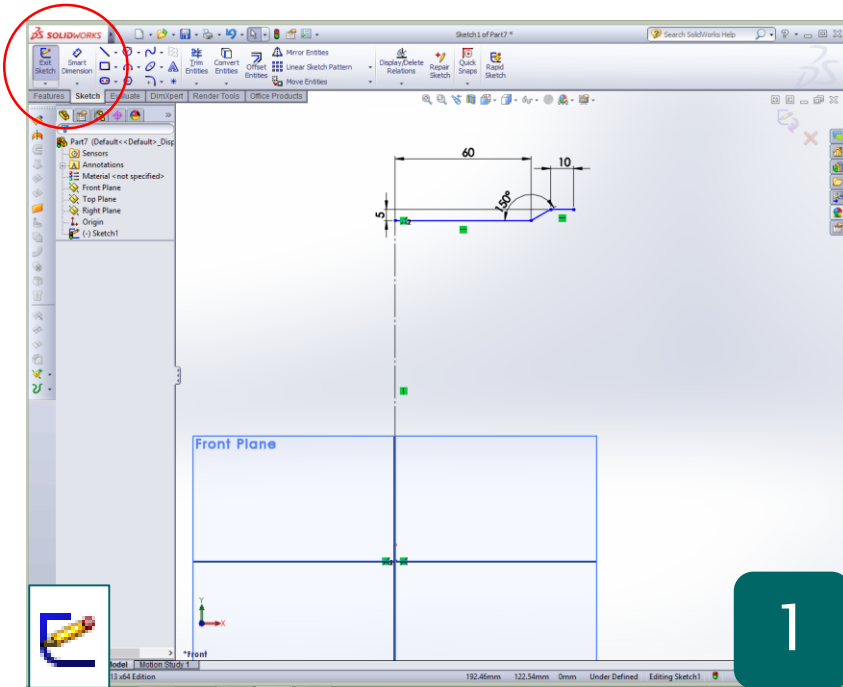
Learning Outcome; Alloy Wheel

Skill Level; 2 - Intermediate

3D; *Boss, Chamfer, Fillet, Revolve, Deform, Point, Cut, Mirror, Circular Pattern*

2D; *pattern, circle tool, mirror, smart dimension, line*

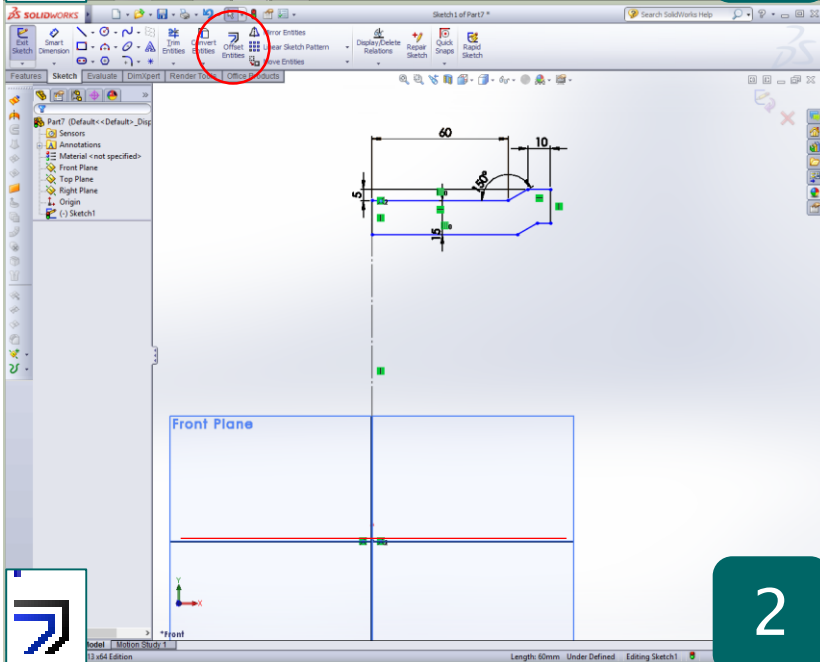




- New > Part File
- 'Show' all planes and select a sketch plane
- Click **sketch** and draw a construction line from the plane intersection
- Dimension; **150mm**
- Replicate the sketch shown using the '**line tool**' and '**smart dimension**'

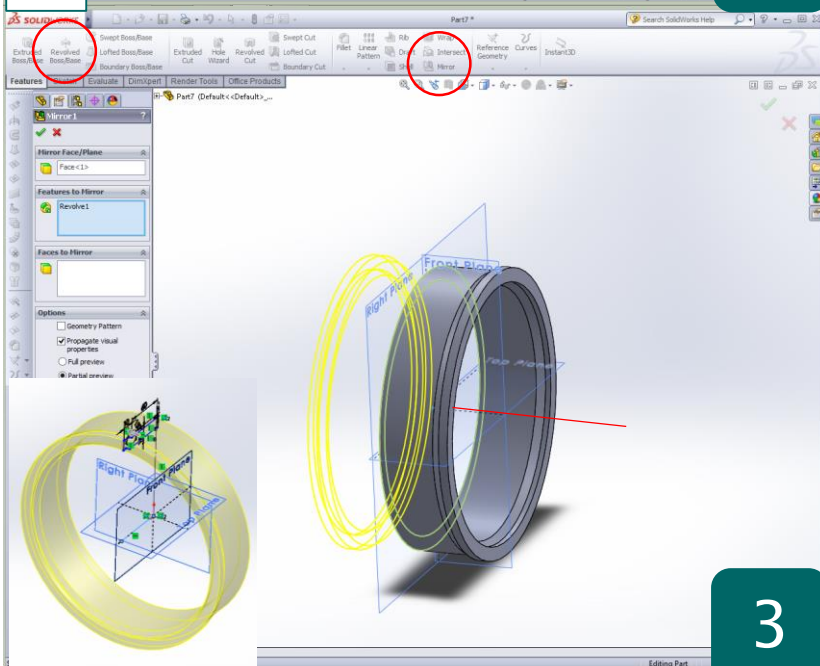
- This tutorial will consider the use of 'Circular pattern' to create complex repeating geometry as well as the 'Deform tool'

Why? – Using this sketch you will create a revolved feature that will form the wheels outer structure. Dimensions are only a rough guide and not representative of a real alloy wheel.



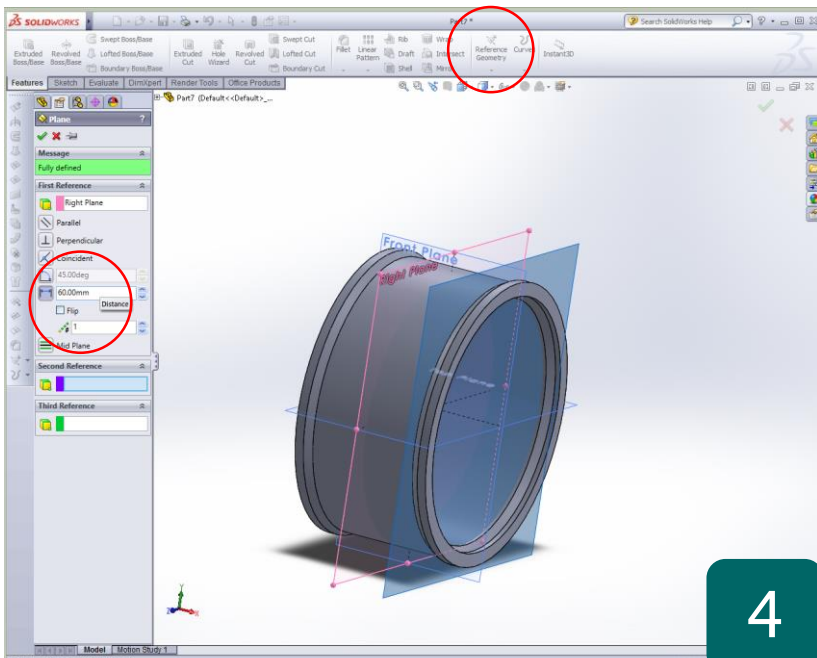
- Drag and Select the completed sketch
- Click '**Offset Entities**'
- Type **15mm** into the dimension box
- Ensure it is going down from the original sketch
- Confirm with the green tick
- Join the ends of each sketch with two lines to ensure a complete profile
- Sketch a **construction line** horizontal at the plane intersection (red line)
- Exit sketch

Why? – To ensure the revolve has the correct thickness the 'offset entities' tool is useful to quickly copy sketch geometry particularly when curves and angle lines are involved



- Features Toolbar
- Select '**Revolved Boss**'
- Select the sketch profile and then the bottom construction line as the '**axis of revolution**'
- Confirm with the green tick if the preview matches the view shown
- Select the '**revolve**' feature in the '**model tree**'
- Click '**mirror**' in the '**features toolbar**'
- Select the work plane as the mirror line as shown

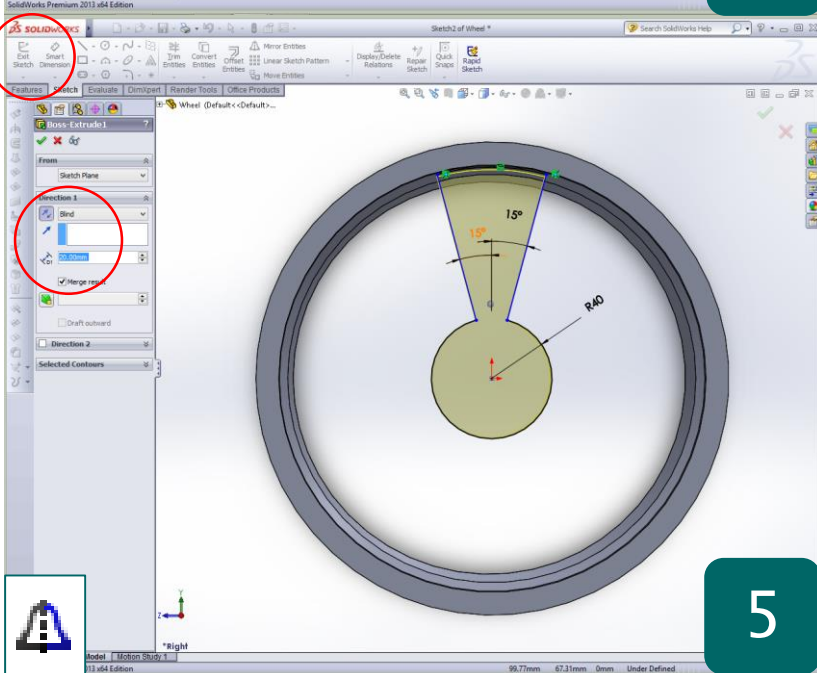
Why? – using the feature 'Mirror' tool can be incredibly useful to create repeating geometry. Multiple features can be selected at once to be mirrored



4

- To create the alloy wheel spoke a new **'reference plane'** is required
- Select **'reference geometry'** in the **'feature toolbar'**
- Click **'Plane'**
- Select the plane at the middle of the model as a reference
- Type **60mm** into the distance setting in the feature menu
- Ensure the preview matches the screenshot
- **'Flip dimension'** if not

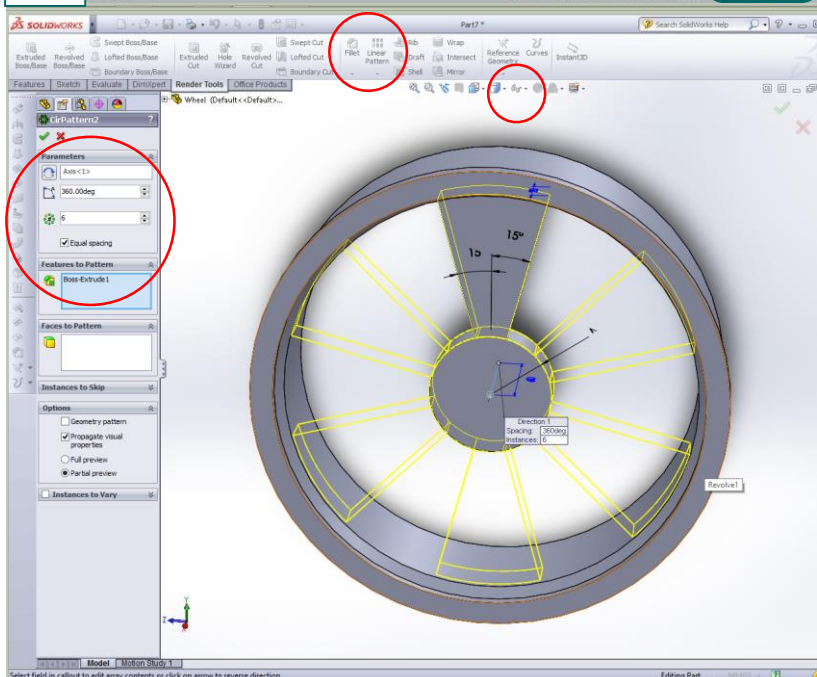
Why? – In a 3D model everything is referenced to set 2D work planes. The three given as default often are not enough to reference sketches and features from and therefore extra ones are created



5

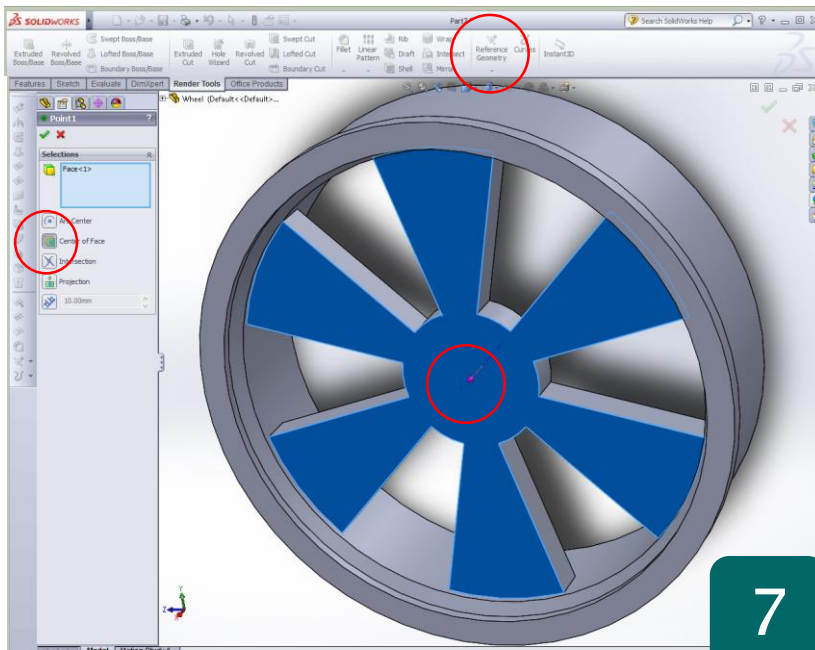
- Select the new **'reference plane'**
- Click **Sketch** and **CTRL + 8** for Normal to
- Replicate the sketch shown using **'Convert Entities'** for the top radius
- **Exit Sketch**
- **'Extruded boss'** into the wheel by **20mm**
- Confirm with the green tick

Why? – **'Convert Entities'** allows you to quickly copy geometry from other parts of the model exactly to prevent any dimensioning errors



- Select the arrow under **'linear pattern'** from the **'feature toolbar'**
- Click on **'Circular pattern'**
- Select the **'extruded boss'** used to create the spoke in the **'features to pattern'**
- Click **'show'** and turn on **'temporary axis'** to select for the rotation axis
- Set the number of instances to **'6'**
- Tick **'Equal Spacing'**
- Confirm with green tick if preview matches the screenshot shown

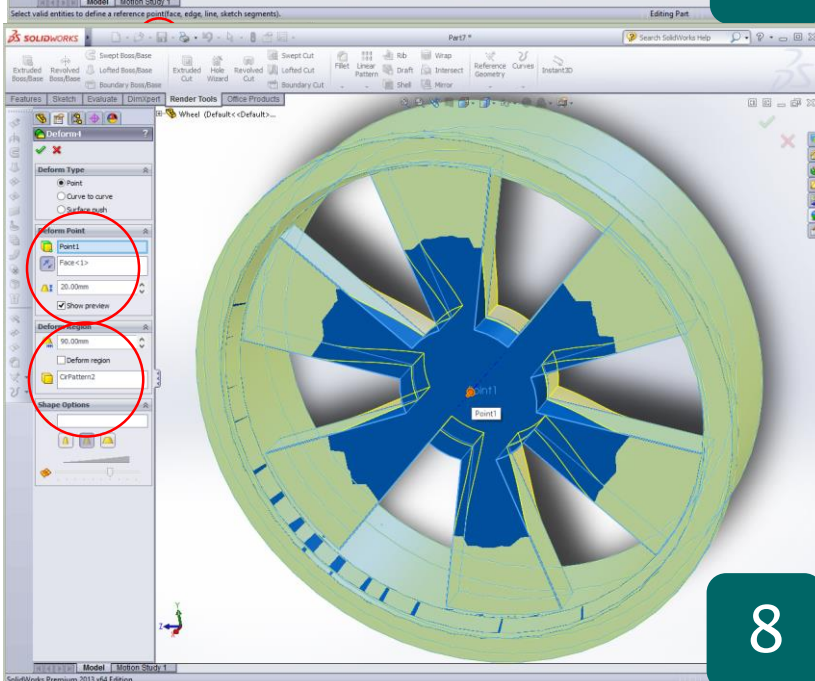
Why? – The **'circular pattern'** feature quickly replicates geometry on a model around a set axis in the same way the sketch circular pattern does in 2D



7

- Select '**reference geometry**' again from the features toolbar
- Click '**Point**'
- Select the point at which the '**temporary axis**' meets the centre face
- Confirm with the green tick

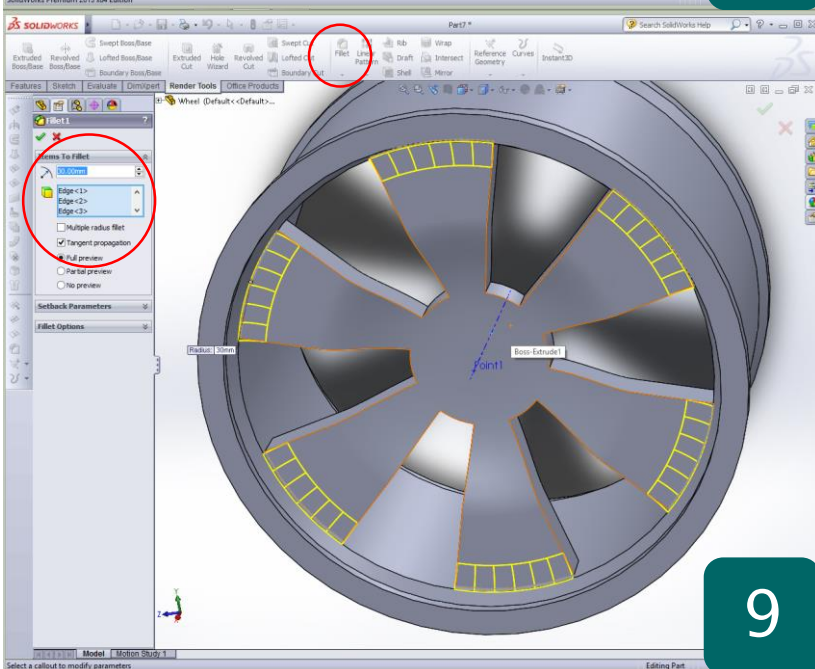
Why? – In order to use the next feature a reference point must first be created. These can be used to access a number of different points on a model to use later in modelling process



8

- Access the '**feature**' menu at the top middle of the screen
- Scroll down to '**features**'
- Select '**Deform**' from the menu
- In feature menu select the newly created 'point' for the '**Deform Point**'
- Set the distance to **20mm**
- Click the front face for deformation
- Set the 'Deform Region' to **90mm**
- Ensure the circular pattern is selected in '**Deform region**'

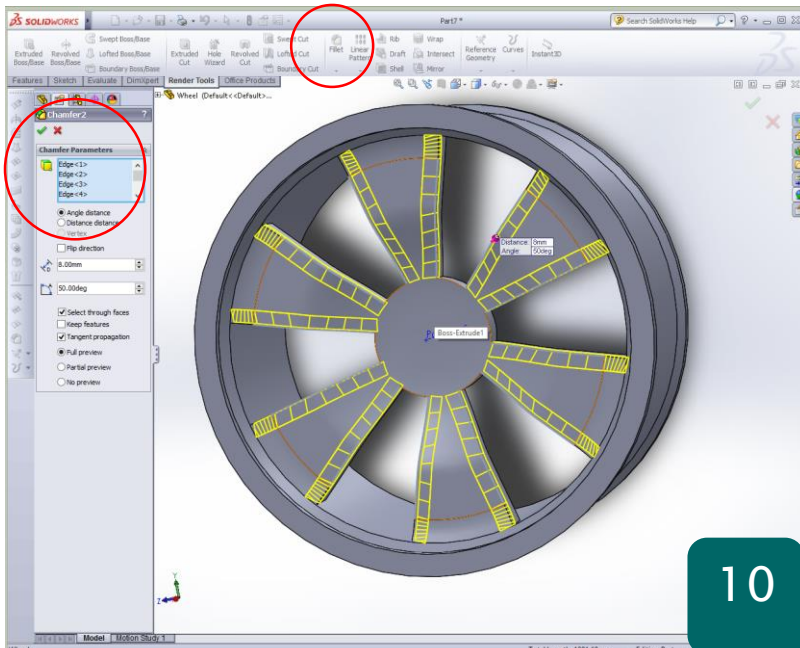
Why? – The Deform Tool creates complex geometry turning the model into a malleable material which can be pushed and pulled. Although accuracy is less this is a quick way to create curved parts



9

- Select '**fillet**' from the features toolbar
- Selecting only lines choose all 6 edges where the spokes meet the wheel arch
- Set the radius to **30mm**
- Confirm with the green tick

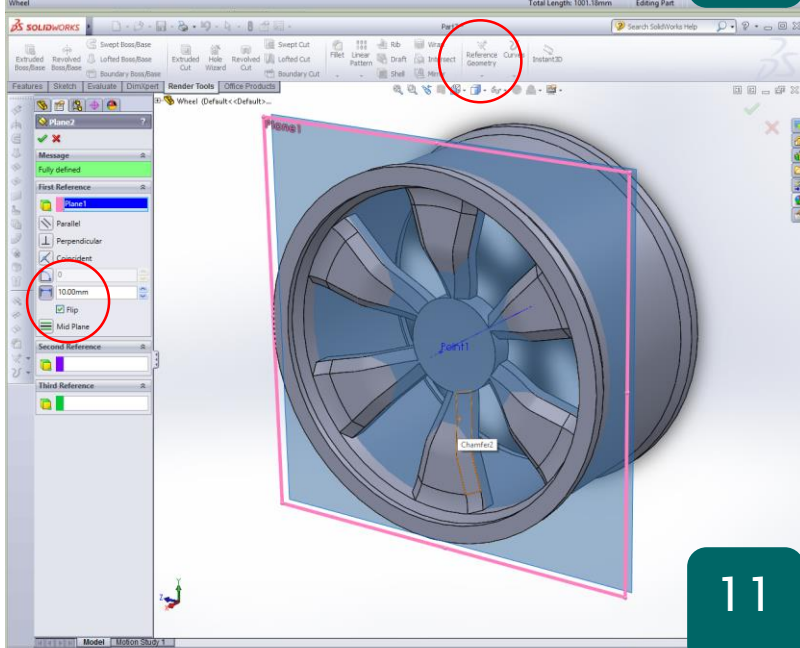
Why? – The use of the fillet tool is a useful way to remove sharp edges that would otherwise not be created in the manufacturing process



10

- Select the 'Chamfer' tool in the 'features toolbar'
- Set the angle to **50°** and depth to **8mm**
- Select all of the long spoke edges around the wheel to achieve the shown preview
- Confirm with the green tick

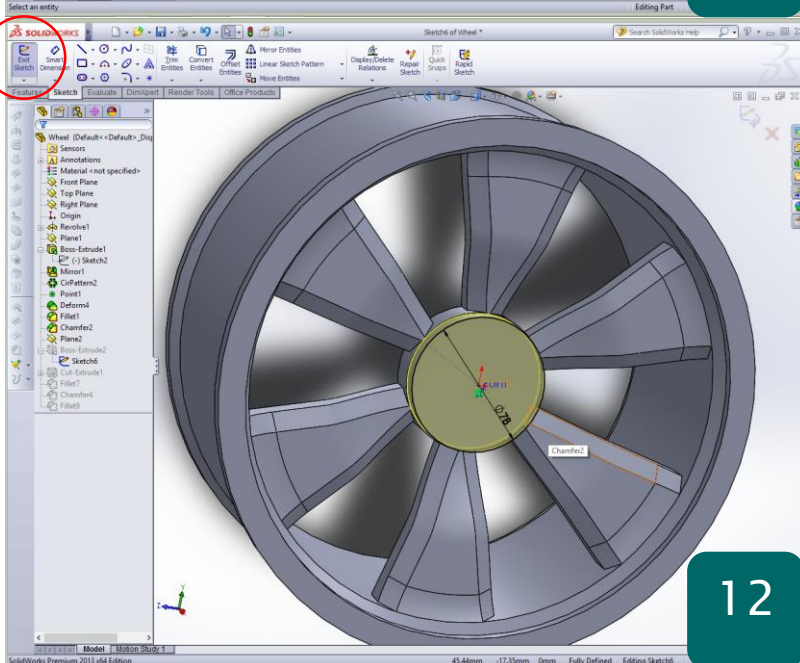
Why? – The use of the chamfer tool gives the alloy wheels a greater realism and adds a reflective face which will show up well in the final render



11

- Select 'Reference Geometry' in the 'features toolbar'
- Select 'Plane'
- Create a plane from the original new plane **10mm** towards the wheel
- Confirm with the green tick

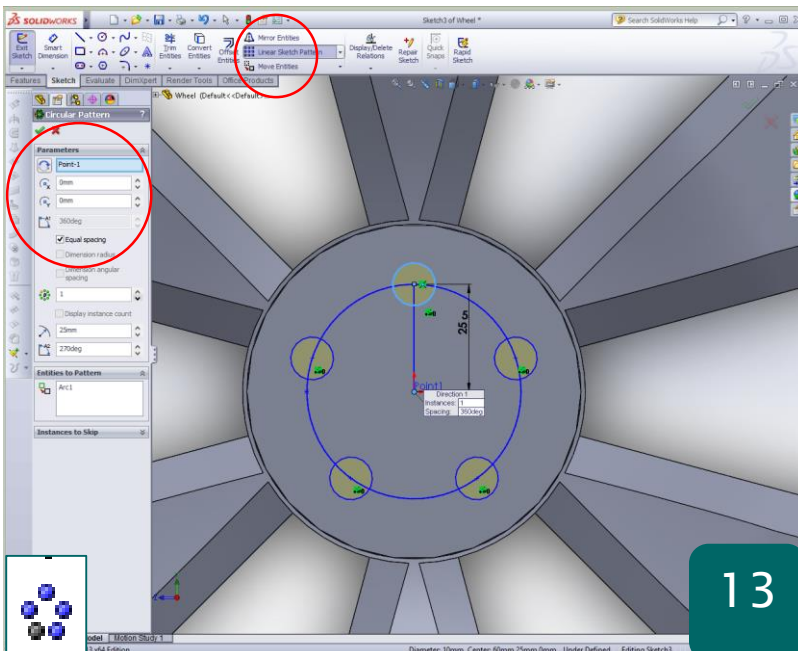
Why? – This reference plane will be used to create the centre of the wheel. Due to the 'Deform' feature the central face is no longer 2D and therefore cannot be sketched on directly



12

- Select the newly created work plane
- Click 'Sketch'
- Sketch a circle with a diameter of **78mm**
- **Extruded boss**
- Set depth to **20mm** into the face of the wheel
- Ensure it goes past the wheel face
- Confirm with the green tick

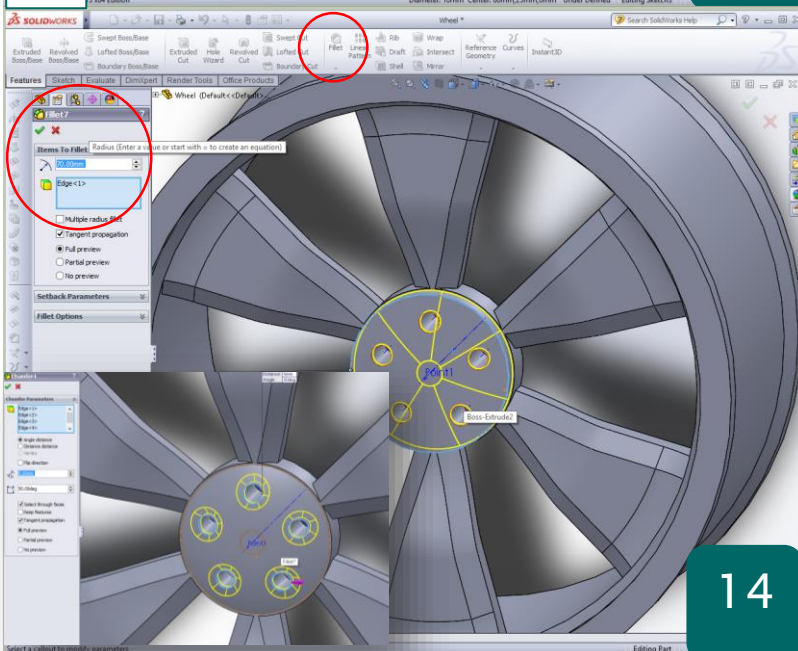
Why? – As the deform tool warped the surface of the wheels centre the extrude must go slightly past the face to ensure it merges fully with the wheel



13

- Select the end face of the extrude
- Click 'Sketch'
- Use CTRL + 8 to bring the view normal
- Sketch a circle 25mm from the centre along the vertical axis
- Click the arrow next to 'Linear Sketch Pattern'
- Select 'Circular patter'
- Set the 'reference point' as the centre
- Instances to '5' and 'equal spacing'

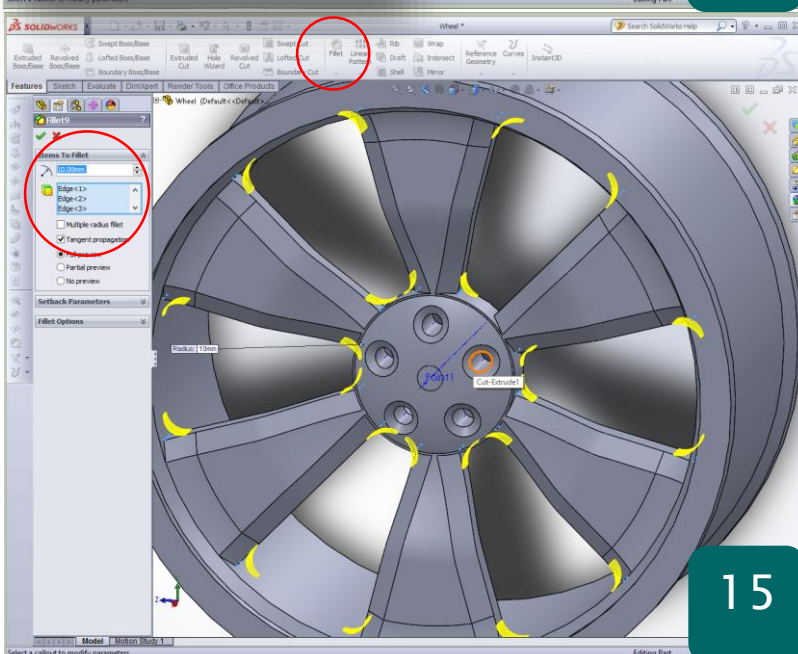
Why? – The circular pattern tool works just as the feature driven tool above but for 2D sketch geometry. This ensures accuracy and saves a great deal of time when dealing with repeating geometry



14

- Access the feature toolbar
- Extruded cut through the wheel and through the other side to create 5 holes
- Confirm with tick
- Select 'Fillet' and round the top edge by 70mm and confirm
- Select 'Chamfer' and chamfer the 5 holes by 5mm

Why? – These finishing steps will give the alloy wheel far greater realism and add to the render finish generated due to increase surface complexity



15

- Select 'Fillet' again from the 'features toolbar'
- Set the radius to 10mm and select all of the inner edges for each wheel spoke
- Continue until preview matches screenshot shown
- Further modifications can be added at this stage if required using similar tools
- Select the 'Edit appearances' ball and apply a 'Chrome' material finish
- Extra colour detail can be added by selecting 'face' in the 'edit appearance' feature menu

Why? – This completes the Alloy wheel tutorial. To further test you skills try modelling a tire to fit onto the alloy in an assembly.