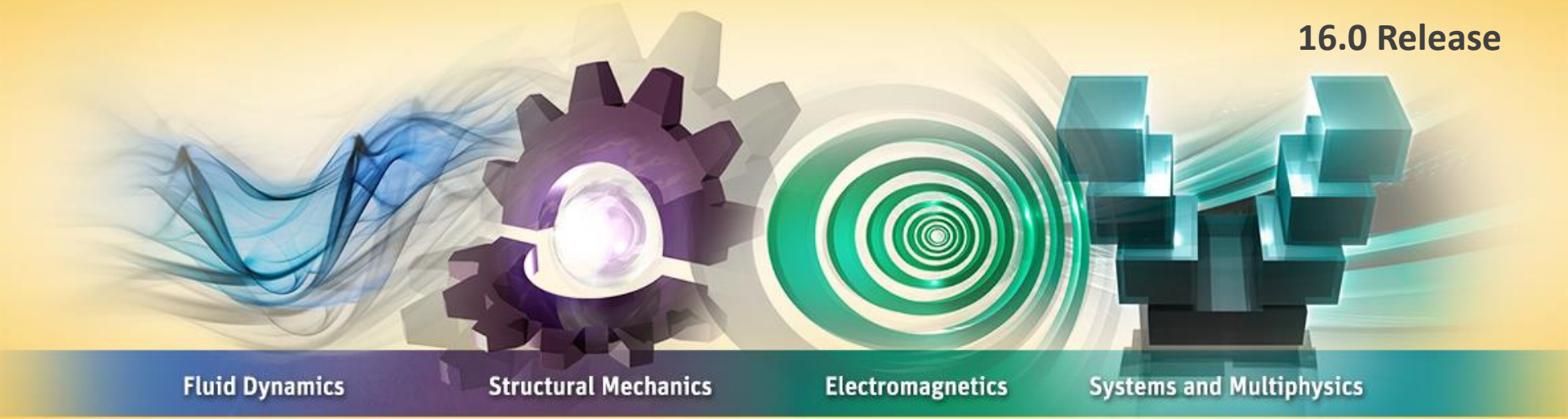
Workshop 3.2
Sketching

16.0 Release



Introduction to ANSYS DesignModeler

Realize Your Product Promise®

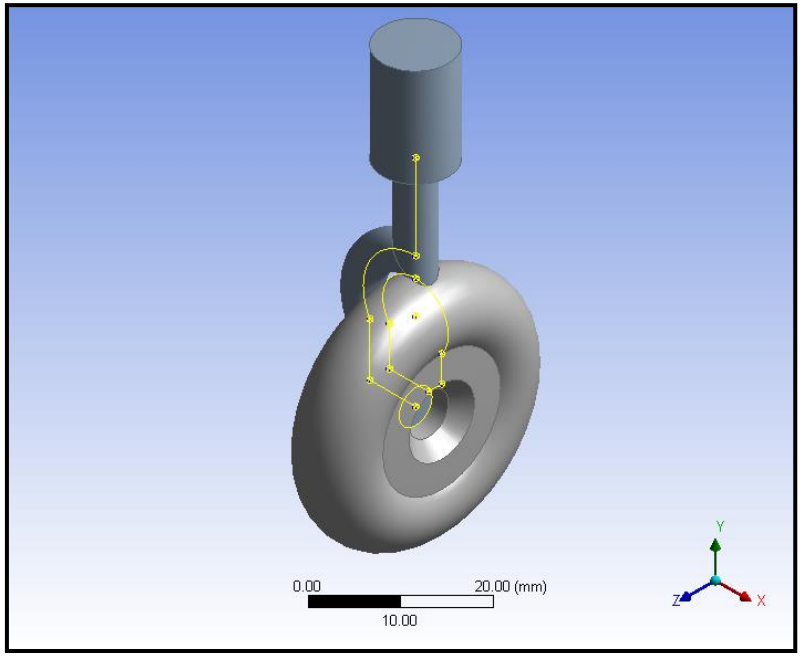
Introduction

Background

- This workshop will introduce further practical sketching techniques. Some simple modelling operations will also be introduced. An example of a simple castoring wheel assembly will be used.
- It is assumed that workshop 1 has been completed.

Objectives

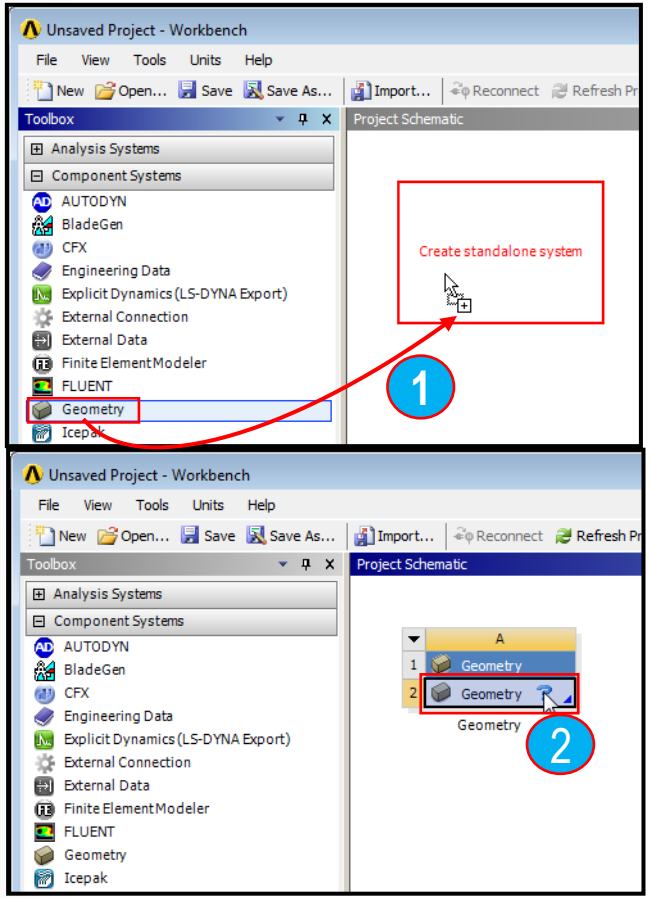
- Create & modify multiple sketches.
- Constraining Sketches.
- Dimensioning Sketches.
- Using Sketches as Base Objects for 3d Operations.



Project Startup

Create the Project

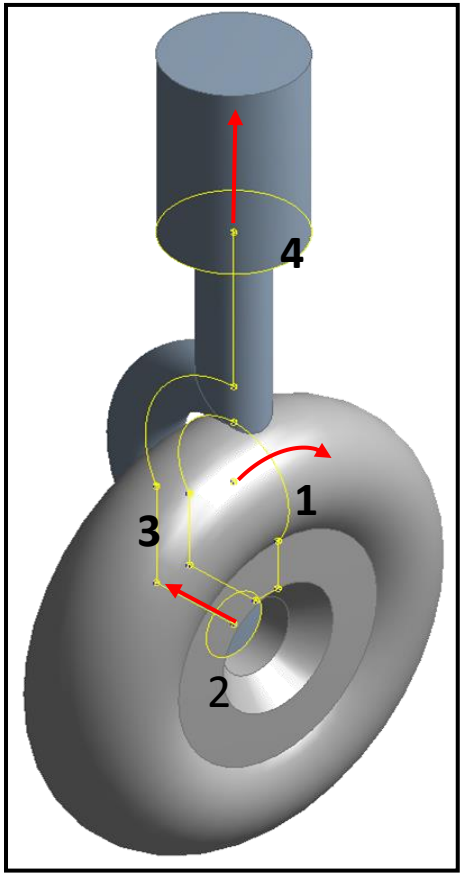
- Start Workbench and follow steps
  1. Drag and drop Geometry component to Project Schematic
  2. Double click on Geometry cell



Preparation

Planning

- Sketch contains
  1. Cross section of the wheel which can be revolved.
  2. Cross section of the shaft which can be swept along a path.
  3. Path that the shaft sketch will be swept along.
  4. Sketch representing the upper cylindrical structure which can be extruded.

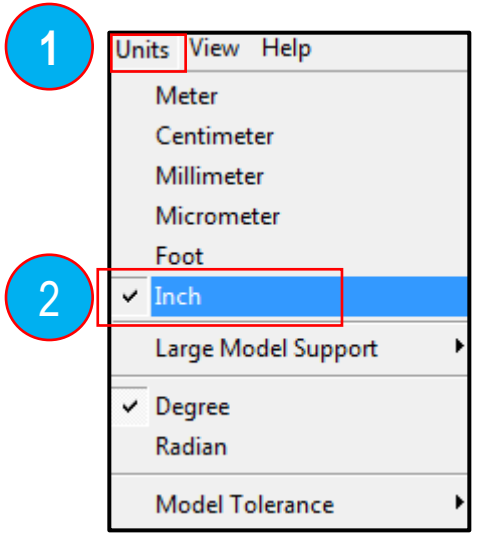


Set Dimensions

Set Units

- **In this session we will be using Inch**

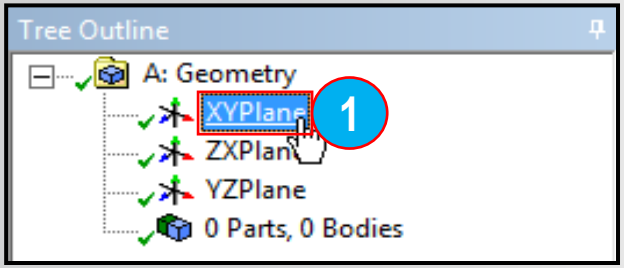
- **Go to 'Units' tab in DesignModeler and select Inch.**
  - ***Units → Inch***



Sketch Creation

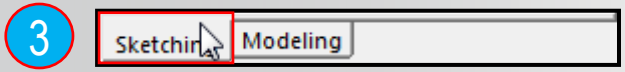
Create a Sketch

- **In the Tree Outline, select the XY Plane.**

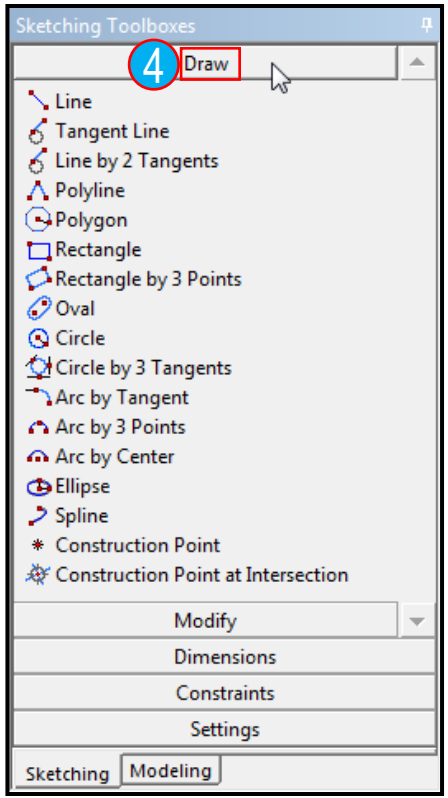


- **Click the Look At button in the toolbar.**

- **Select the Sketching Mode Tab.**

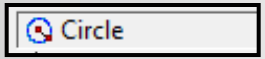


- **Select the Draw Toolbox.**

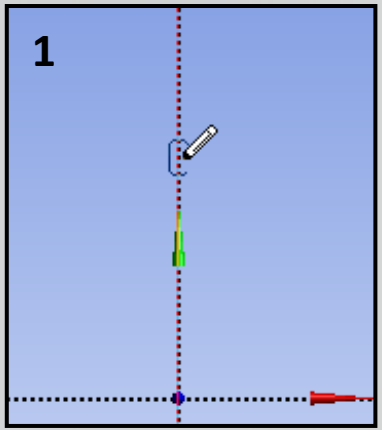


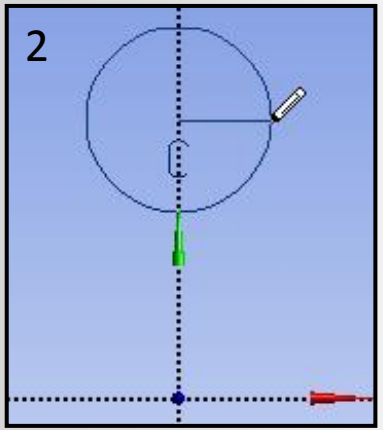
Drawing a Circle

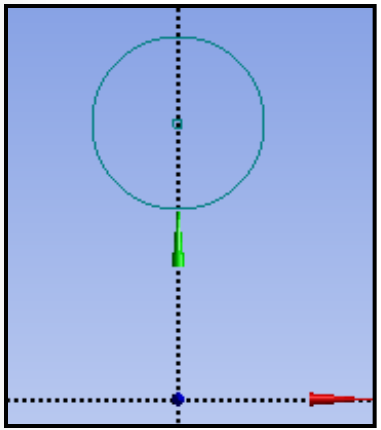
Create a Sketch (Continued)

- Select the Circle Tool 

- Move the cursor over the Y Axis until a letter ‘C’ appears indicating a coincident auto constraint then click once
- Move the cursor away from the origin to define the radius approximately as shown and click once again







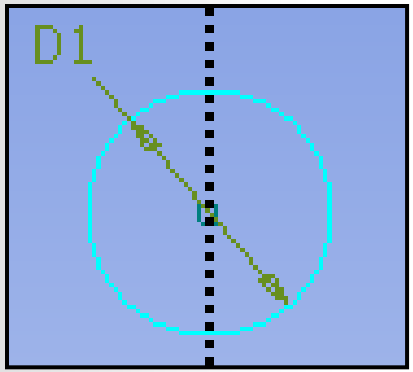
Dimensioning a Circle

Adding Dimensions

- Select the Dimensions Toolbox.
- Select the General Tool. 

- Click the circle edge and click to position the dimension.
- Set the dimension to 10 inch in the Details View.

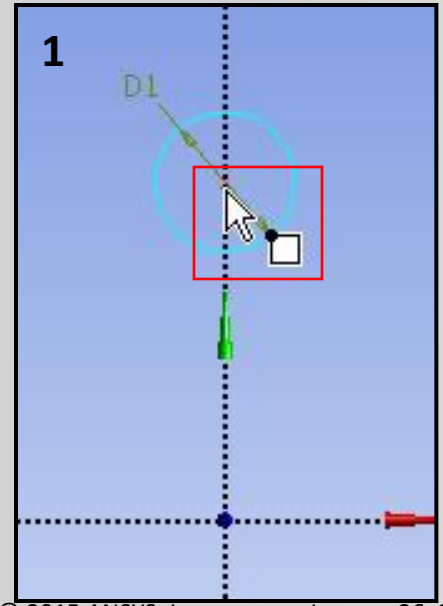
Details View	
Details of Sketch1	
Sketch	Sketch1
Sketch Visibility	Show Sketch
Show Constraints?	No
Dimensions: 1	
D1	10 in
Edges: 1	
Full Circle	Cr7

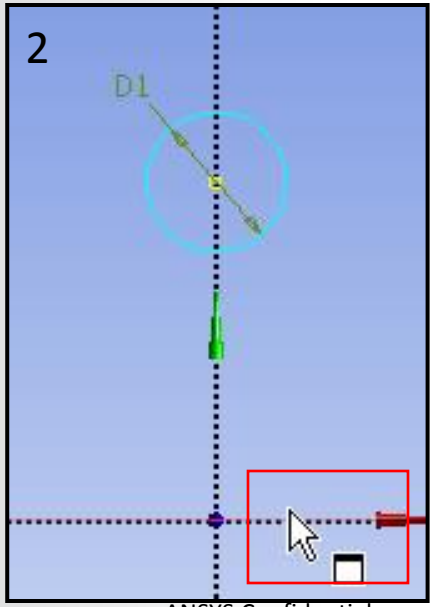


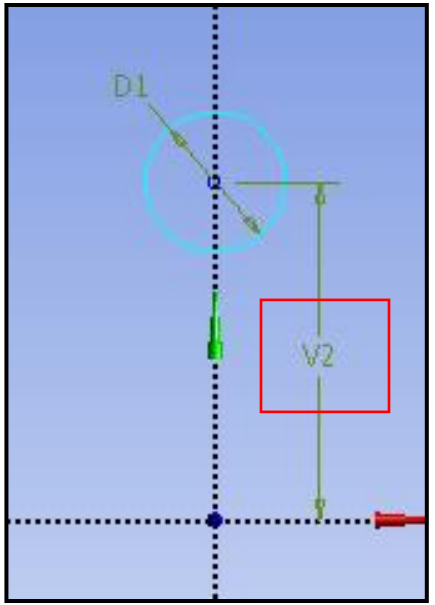
Dimensioning

Adding Dimensions

- Click the centre point of the circle, click the X Axis and position the dimension to define a vertical dimension for the circle centre as shown in the sequence below.





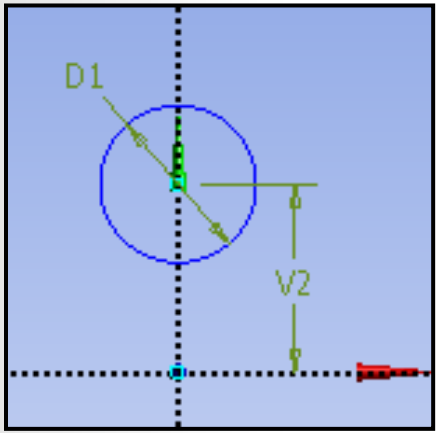


Dimensioning

Adding Dimensions

- In the Details View set the new vertical dimension to 12inch
- Now some dimensions have been established you may wish to adjust the view.
- Click the Zoom to Fit button.

Details View	
Details of Sketch1	
Sketch	Sketch1
Sketch Visibility	Show Sketch
Show Constraints?	No
Dimensions: 2	
D1	10 in
V2	12 in
Edges: 1	
Full Circle	Cr7

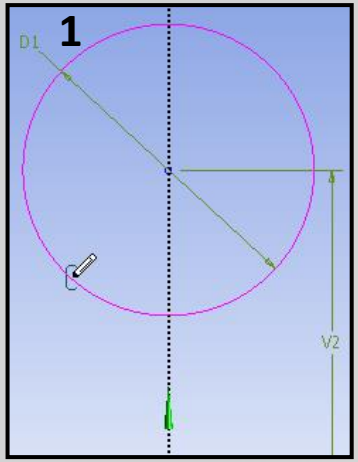


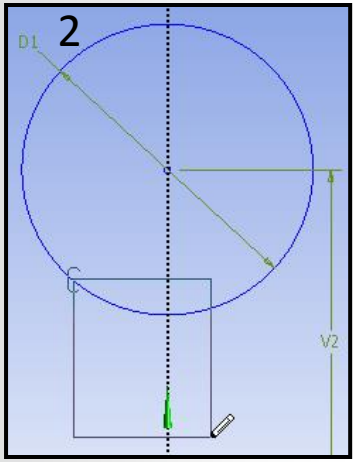
Rectangle

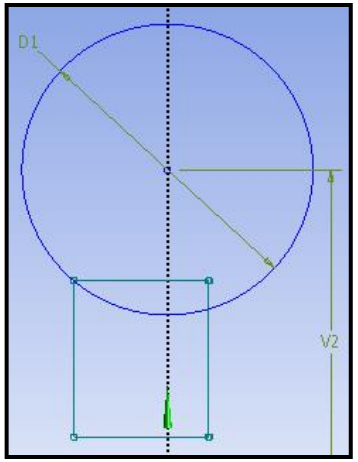
Draw a Rectangle

- In the Draw Toolbox select the Rectangle Tool 

- Click on the left side of the circle (make sure you see the C coincident constraint)
- Move the cursor diagonally down and to the right clicking again in the approximate position shown.



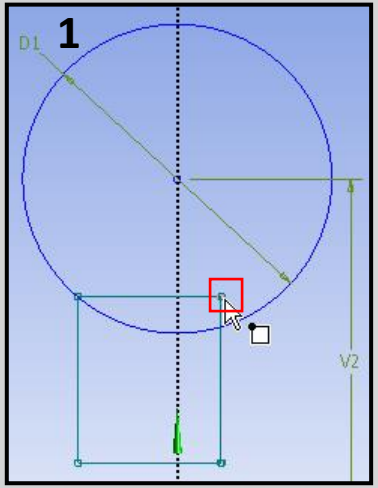


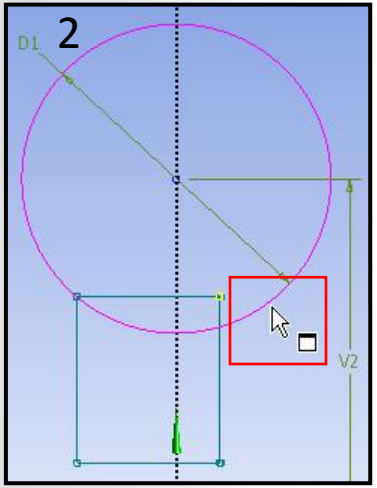


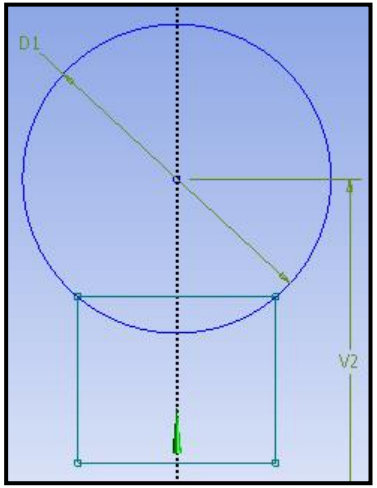
Adding a Coincident Constraint

Add a Coincident Constraint

- **In the Constraints Toolbox select the Coincident Tool.**
- **Select the upper right vertex of the rectangle followed by the circle edge as shown in the sequence below.**
  - **The rectangle will snap to the circle edge.**



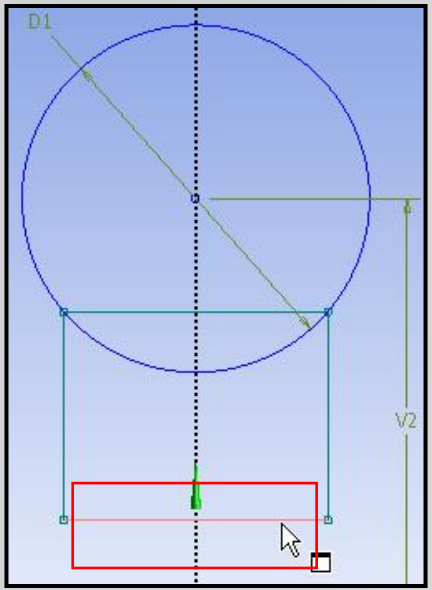


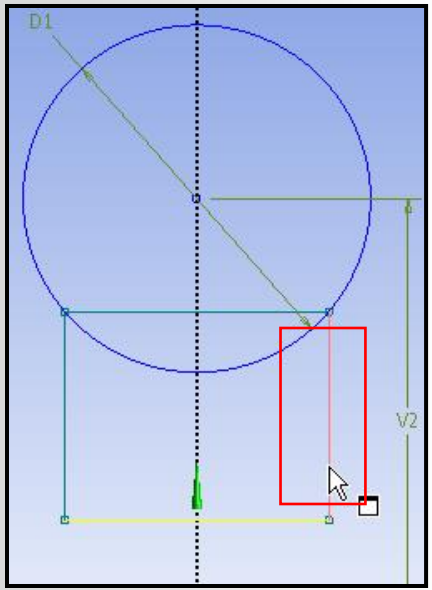


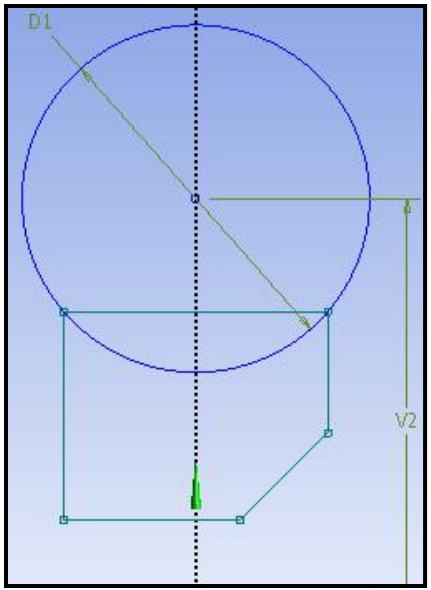
Chamfer

Create a Chamfer 

- **In the Modify Toolbox select the Chamfer Tool, use a chamfer length of 2.5inch.**
- **Click the two edges in sequence as shown below to create the chamfer.**



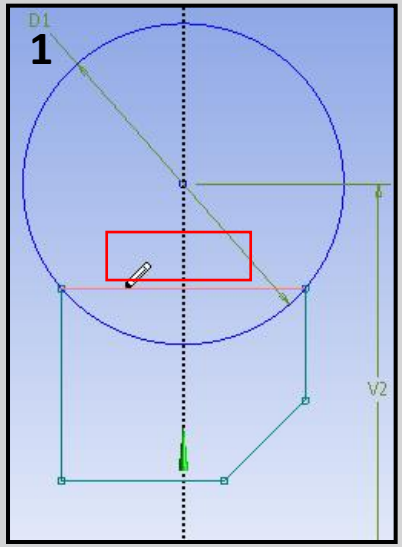


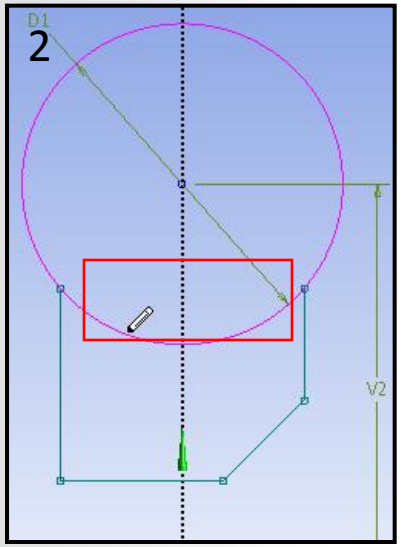


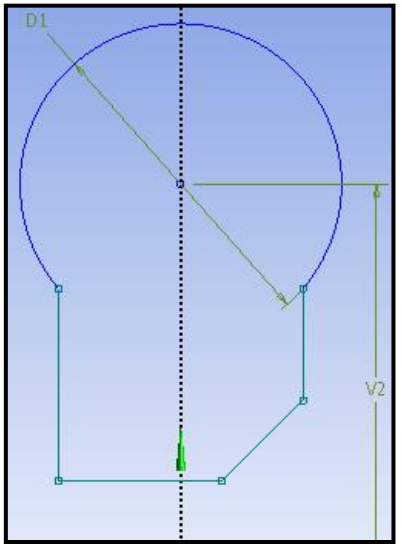
Trimming

Trim Unwanted Edges

- In the Modify Toolbox select the Trim Tool, select the Ignore Axis option.
- Click the two edges in sequence as shown. The edges will be trimmed to the next connected entity ignoring the axis.



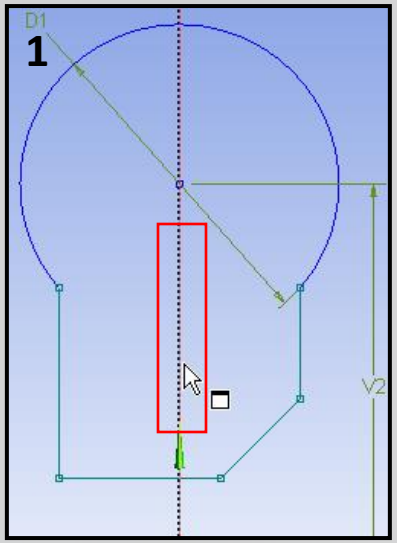


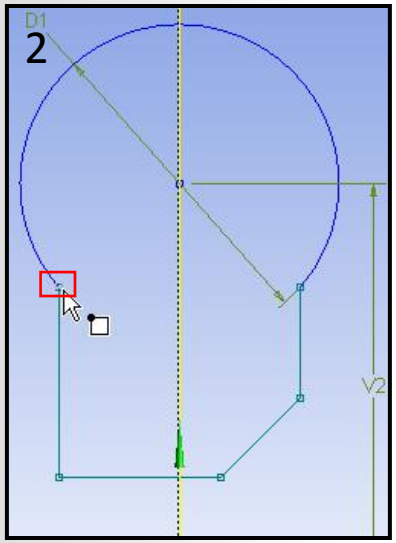


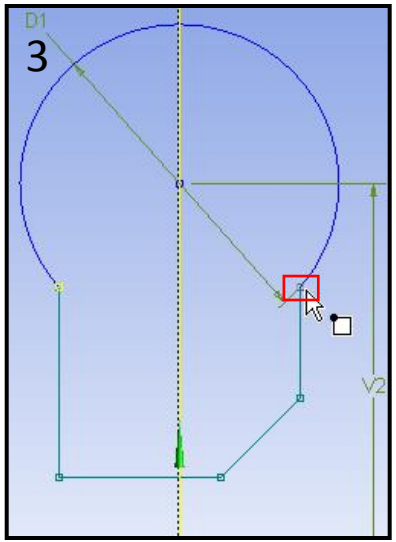
Adding a Symmetry Constraint

Add a Symmetry Constraint

- **In the Constraints Toolbox select the Symmetry Tool.**
- **Select the Y Axis, the point on the left then the point on the right as shown in the sequence below.**





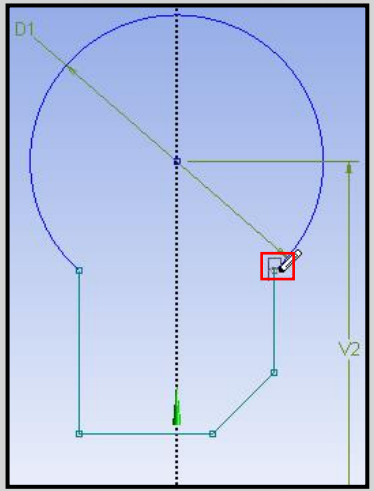


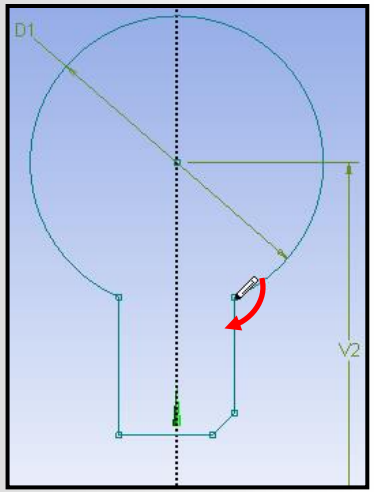
Drag

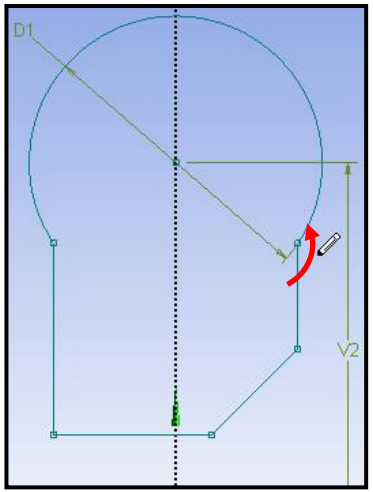
Review Constraints

- From the Modify Toolbox select the Drag Tool. 

- Click and hold on the point as shown
- Drag up and down - you can see that the two points are constrained as symmetric.
- Release then click the undo button to restore the original position. 





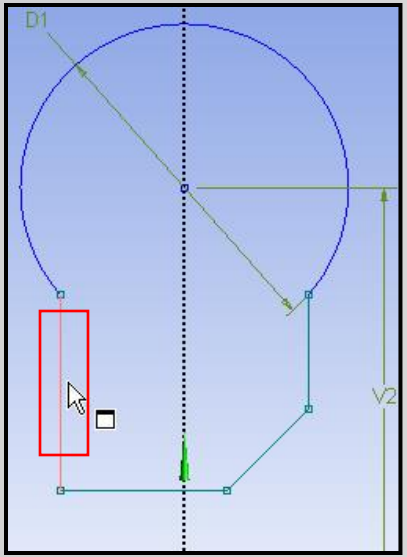


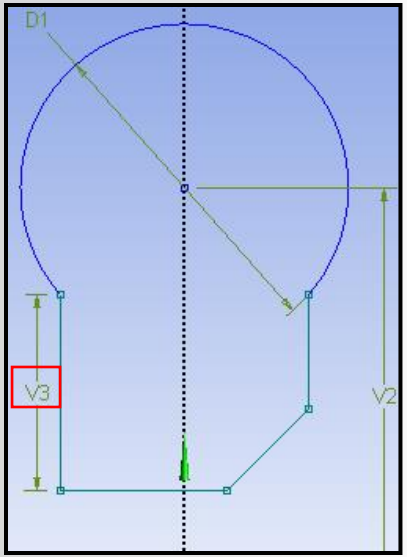
Dimensioning

Add Dimensions

- From the Dimensions Toolbox select the General Tool. 

- Select the edge to apply a vertical dimension as shown.
- In the Details View set the dimension to 6inch.





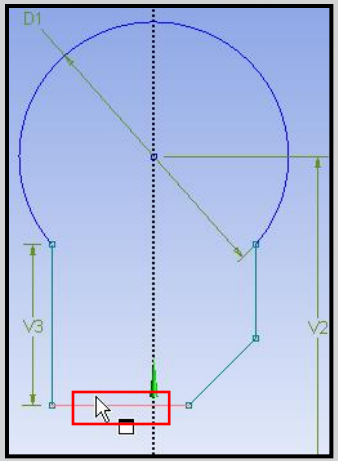
Details View

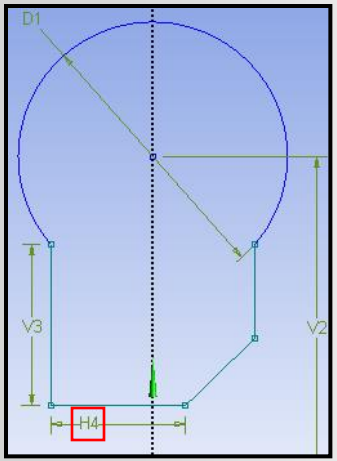
Details of Sketch1	
Sketch	Sketch1
Sketch Visibility	Show Sketch
Show Constraints?	No
Dimensions: 3	
D1	10 in
V2	12 in
V3	6 in

Dimensioning

Add Dimensions

- Select the edge to apply a horizontal dimension as shown using the General Tool again.
- In the Details View set the dimension to 5inch.





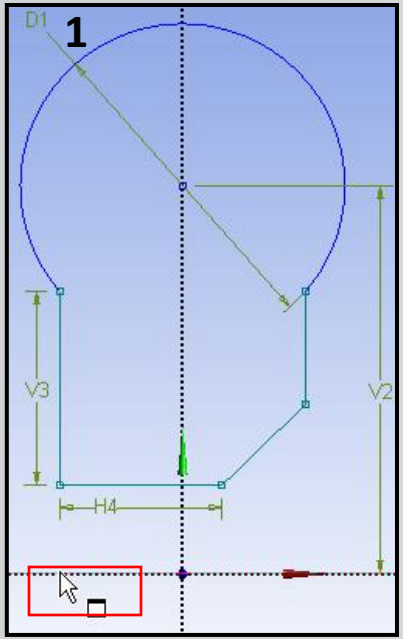
Details View

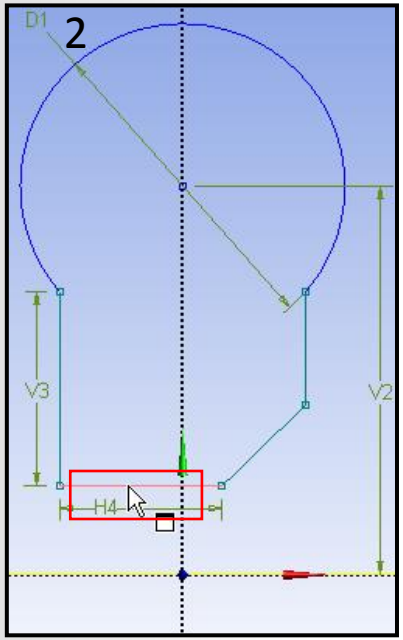
Details of Sketch1	
Sketch	Sketch1
Sketch Visibility	Show Sketch
Show Constraints?	No
Dimensions: 4	
D1	10 in
H4	5 in
V2	12 in
V3	6 in

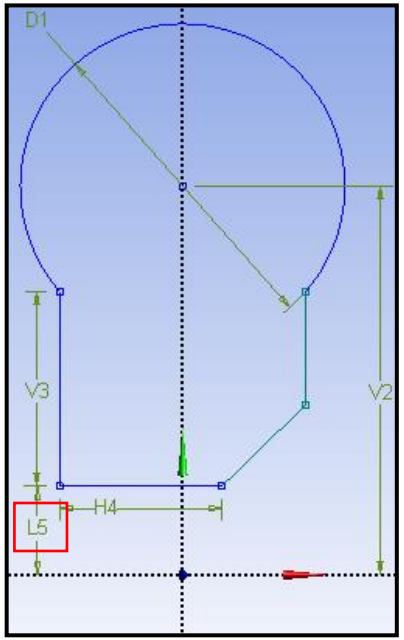
Dimensioning

Add Dimensions

- Select the X Axis then the edge as shown in the sequence below to apply a length dimension using the General Tool setting the value to 3inch in the Details View.



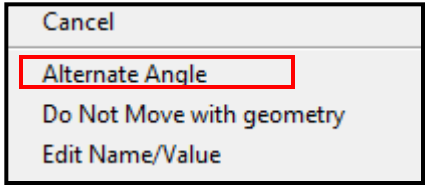


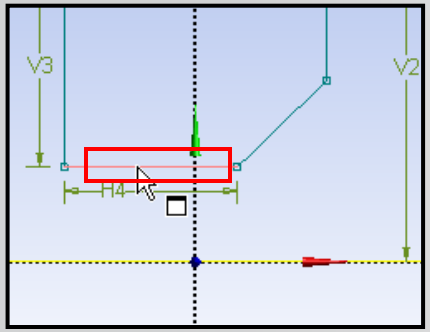


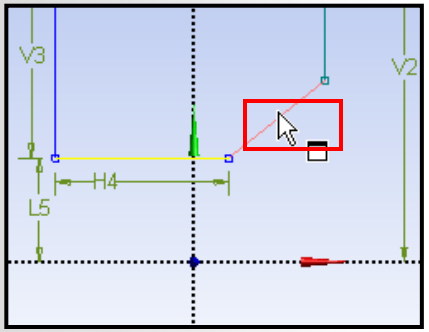
Angle Dimension

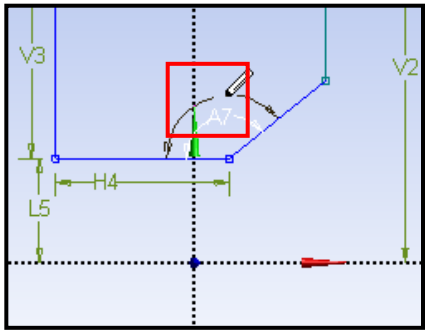
Add Dimensions

- Select the two edges in sequence as shown below using the 'Angle' tool in the 'dimensions' tab.
- A preview of the suggested angle measurement will be shown - right click and select alternate angle from the context menu which appears - repeat until the correct angle definition is displayed as shown.
- In the Details View set the new angle dimension to 135.

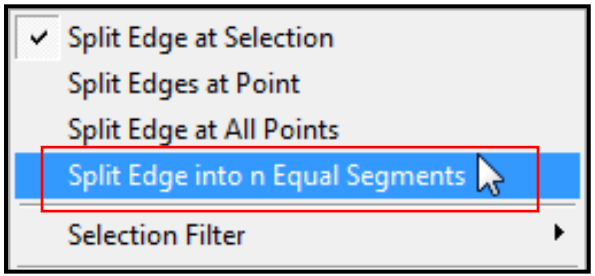






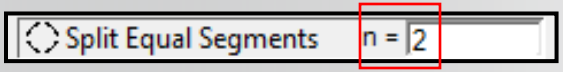


Split

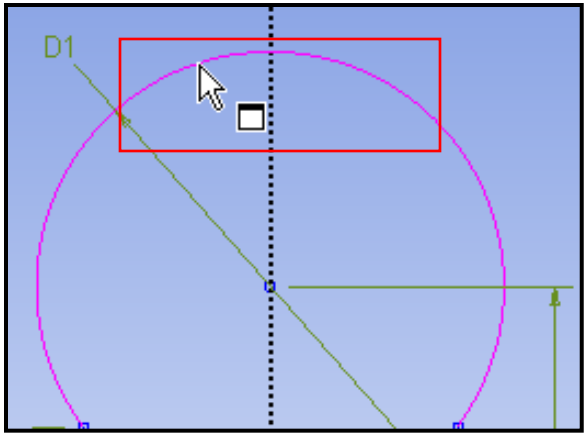


Split Edges

- From the modify Toolbox select the Split Tool.
- Right click in the Graphics Window and select Split into n Equal Segments.
- In the Modify Toolbox set the split parameter to 2.



- Select the curved edge.
  - The edge will be split into two equal lengths.

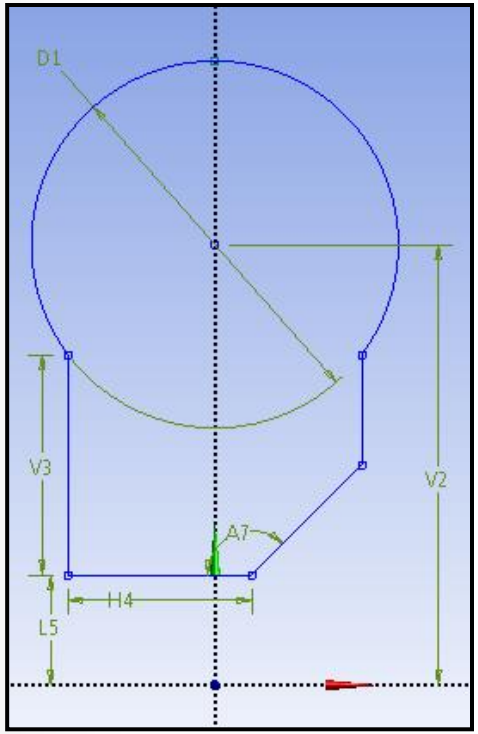


Constraint Status

Add Dimensions

- **All entities in the sketch are blue indicating that the sketch is fully constrained.**
- **At this stage check your sketch matches the image shown here.**

- **We can now apply a 3d Operation to this sketch to create the wheel geometry.**

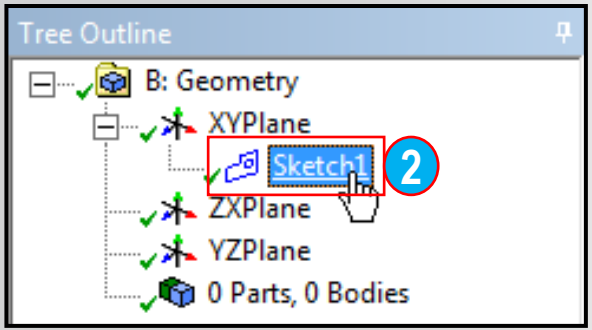


Revolve

Revolve the Sketch

- **Switch to Modeling Mode using the tab.**

- **Select Sketch1 from the Outline.**

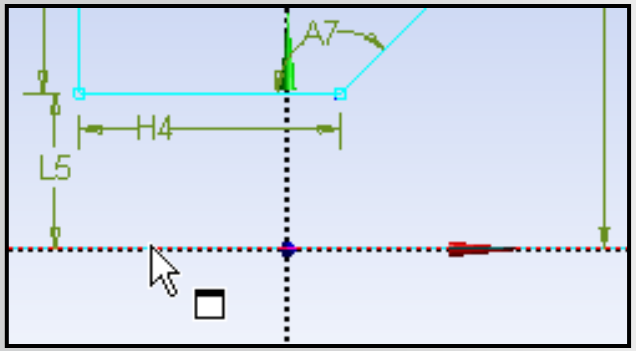


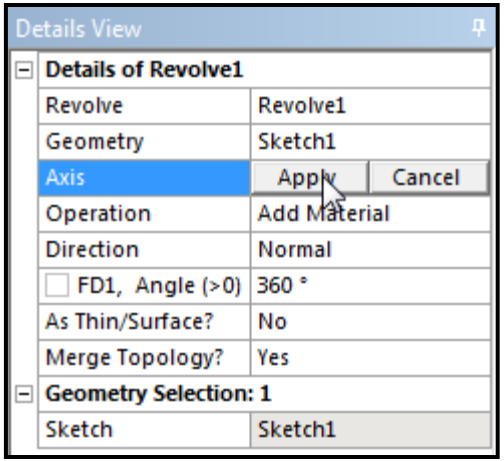
- **Select Revolve from the Create menu or the main toolbar.**

Revolve

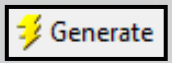
Revolve the Sketch (Continued)

- In Details View, click in the right column of 'Axis' Selection row to activate it. Select the X Axis in the Graphics Window and click Apply back in the Axis Selection Box.





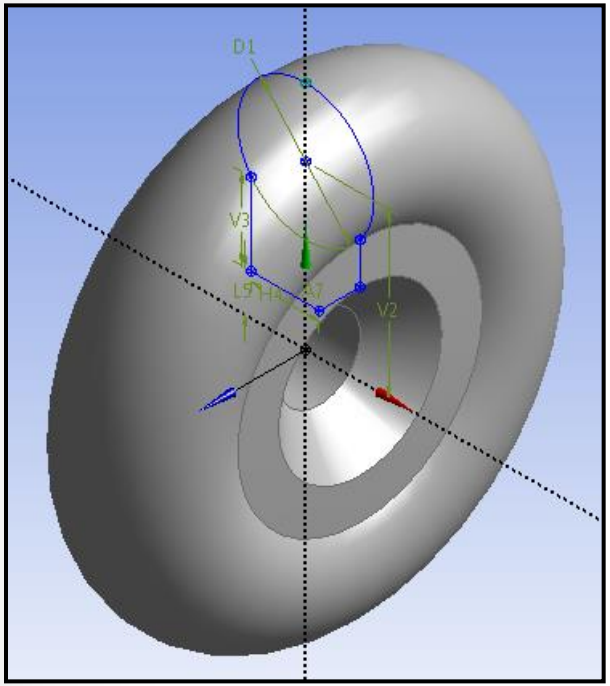
- Click Generate.



Review

Review the Model

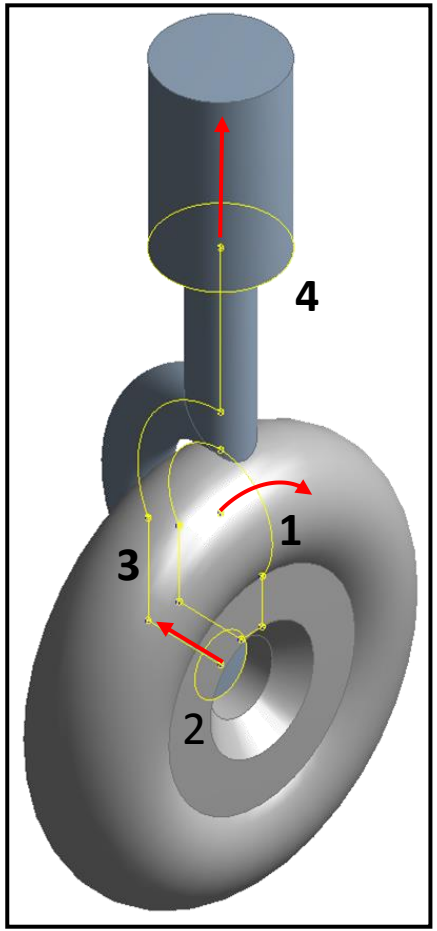
- **Having completed the first sketch and applied a simple modeling operation the first stage of the geometry is completed.**



Preparation

Planning

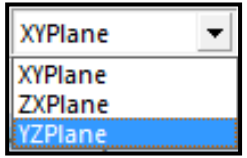
- **The next stage will be the creation of the simple circular sketch (2) which will be used to form the shaft and leg of the geometry.**

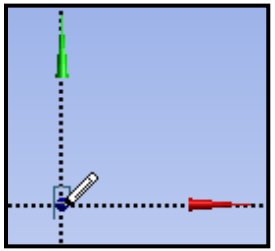


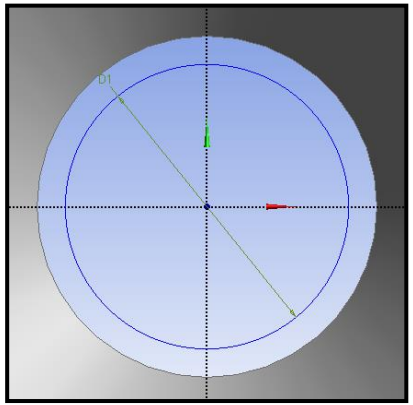
Sketch Creation

Draw and Dimension a Circle

- Switch to Sketching Mode using the tab.
- Select the YZPlane from the drop down box, the view will automatically orientate
- Zoom into the centre as shown.
- Using the Circle Tool in the Draw Toolbox draw a circle centred on the origin. (P = Point Constraint)
- Create a Diameter dimension using the General Tool in the Dimensions Toolbox and set it to 5inch in the Details View.



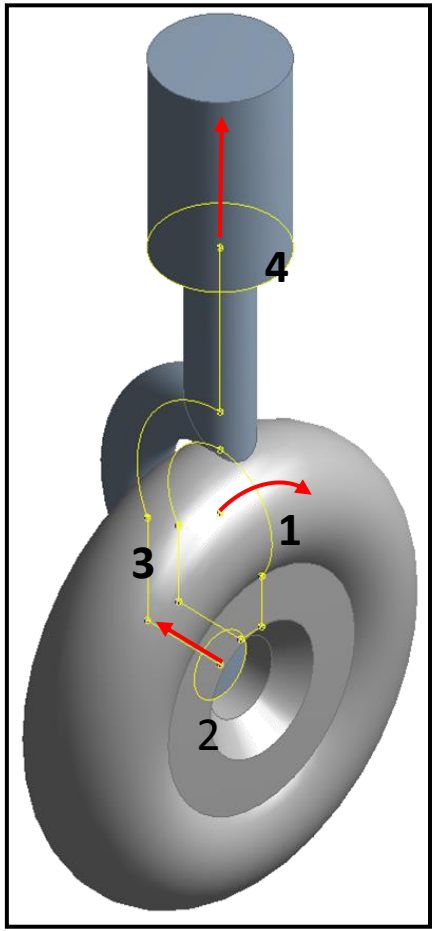




Preparation

Planning

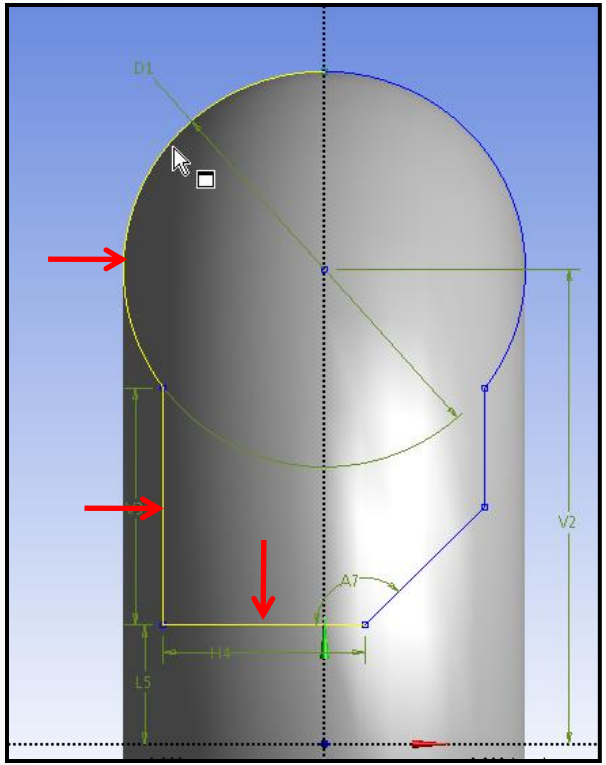
- **Now we'll create the sketch (3) containing the path which will be used to sweep the circular sketch we have just created.**



Sketch Creation

Create an Offset

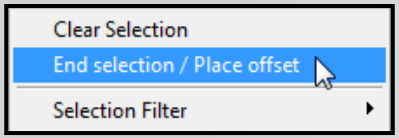
- Select the XYPlane from drop down box.
- Click the new sketch button. 
- From the Modify Toolbox select the Offset Tool.
- Select the three edges highlighted in the image (hold down CTRL to multiple select).



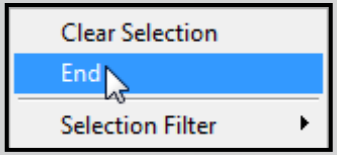
Offset

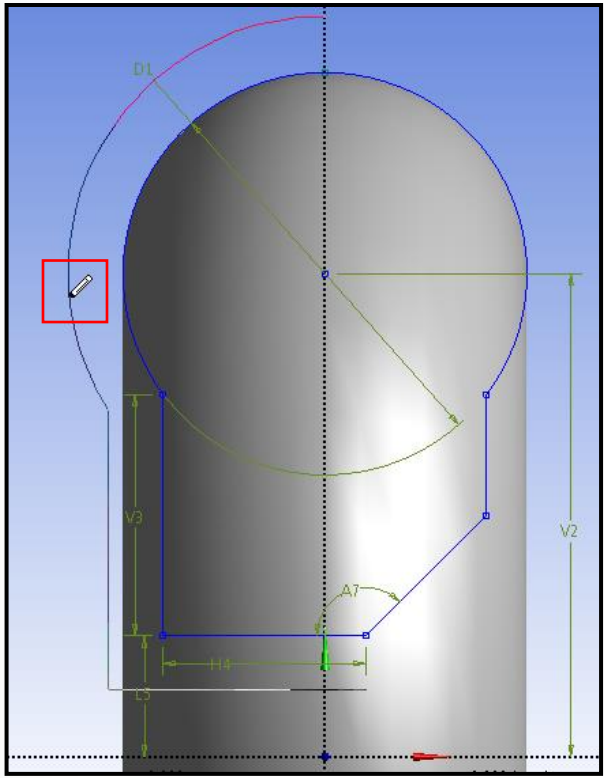
Create an Offset (Cont...)

- Right click and select End selection / Place offset from the context menu.



- The offset location can be selected by moving the cursor.
- Position it approximately as shown and click once.
- Exit the Offset Tool by right clicking and selecting End from the context menu.



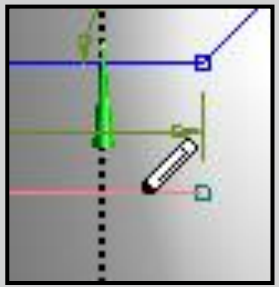


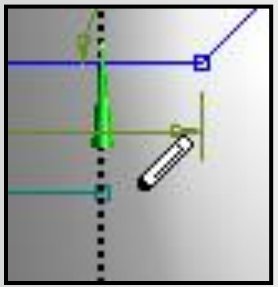
Trim

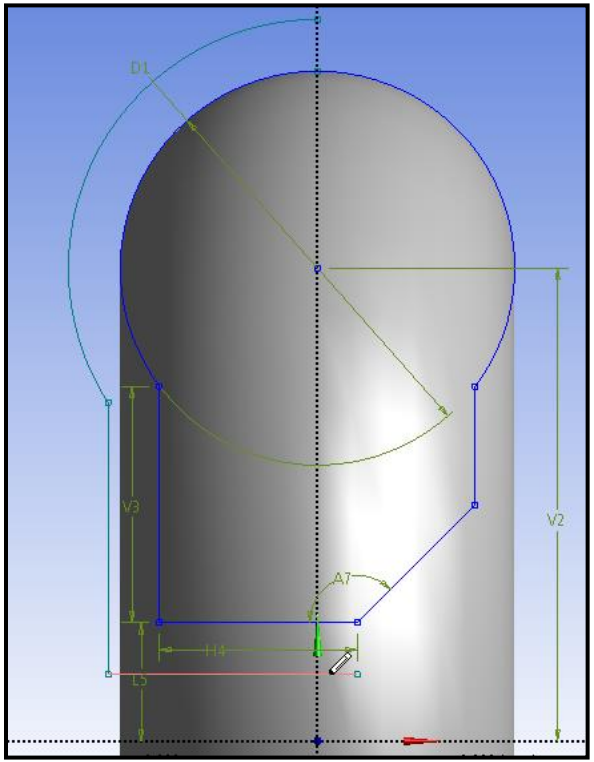
Trim the Sketch

- In the Modify Toolbox select the Trim Tool – this time uncheck the Ignore Axis box.

- Select the portion of the edge to the right of the axis as shown (enlarged below) to trim the edge to the axis.



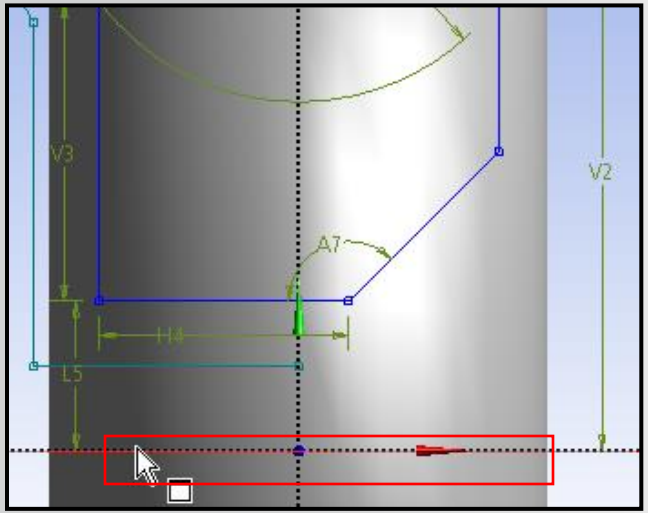


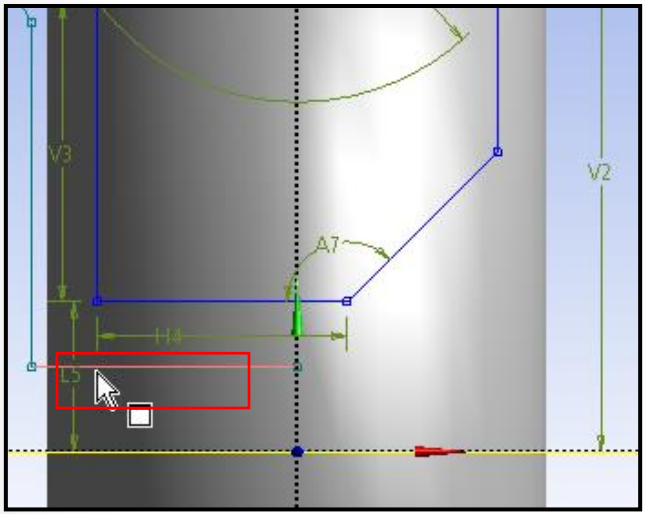


Constraints

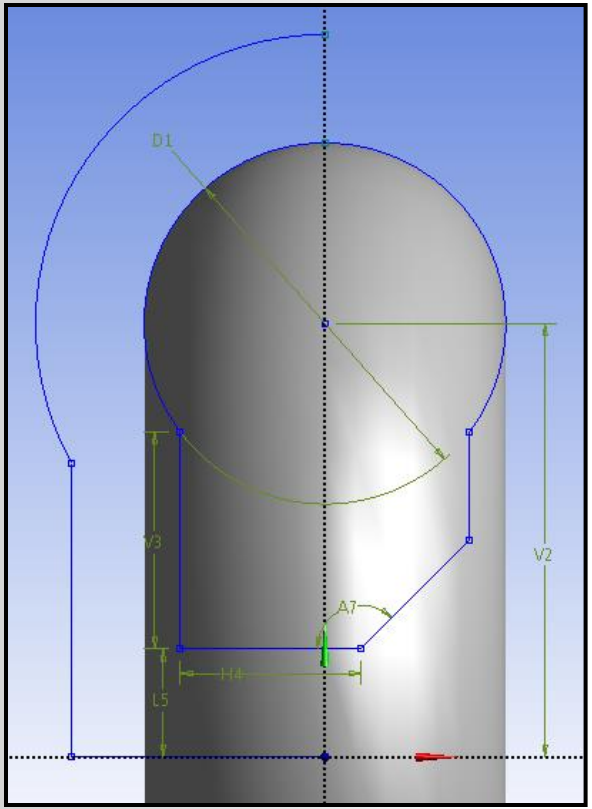
Constrain the Offset

- From the Constraints Toolbox select the Coincident Tool.
- Select the X Axis.
- Select the Edge on the offset sketch as shown.
- The offset sketch will snap to the X Axis (image on next slide)





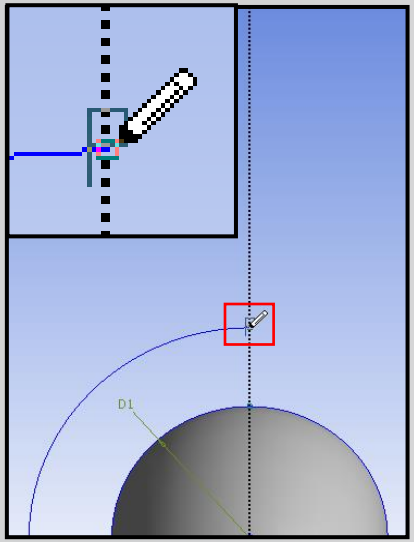
Review

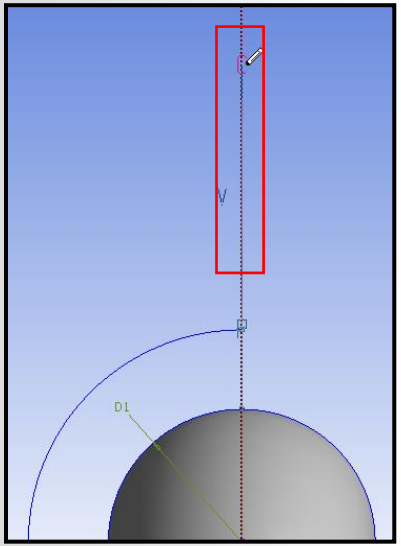


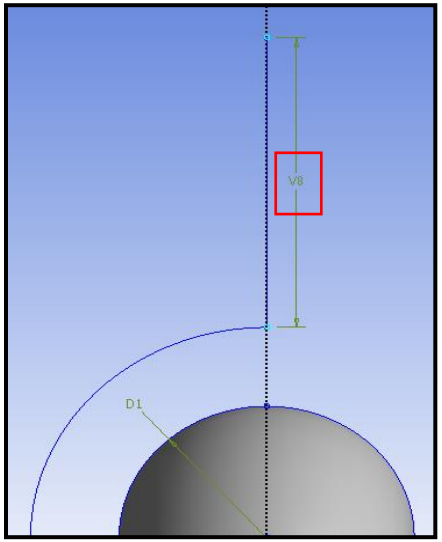
Line

Add a Line

- From the Draw Toolbox select the Line Tool.
- Select the point at the top of the arc and select a point on the Y-Axis in the approximate position shown (zoom out first).
- Use the General Dimension Tool to add a vertical dimension to the line and set it to 13inch.



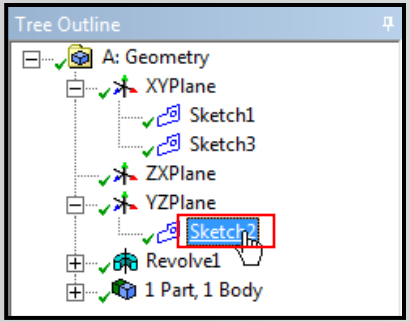




Sweep

Create a Sketch (Continued)

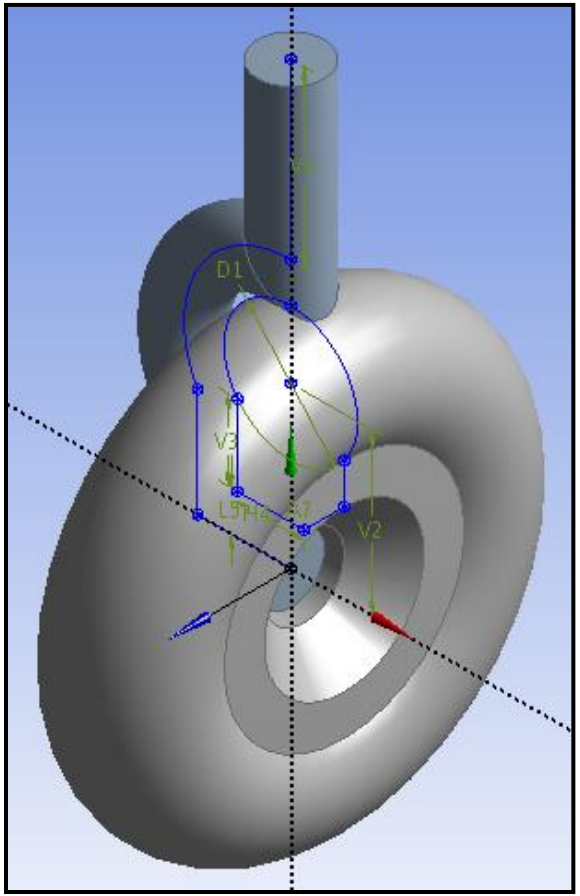
- Switch to Modeling Mode using the tab.
- Select Sketch2 from the Outline.



Details of Sweep1	
Sweep	Sweep1
Profile	Sketch2
Path	Sketch3
Operation	Add Material
Alignment	Path Tangent
FD4, Scale (>0)	1
Twist Specification	No Twist
As Thin/Surface?	No
Merge Topology?	No
Profile: 1	
Sketch	Sketch2

- Select Sweep from the Create menu or main toolbar. 

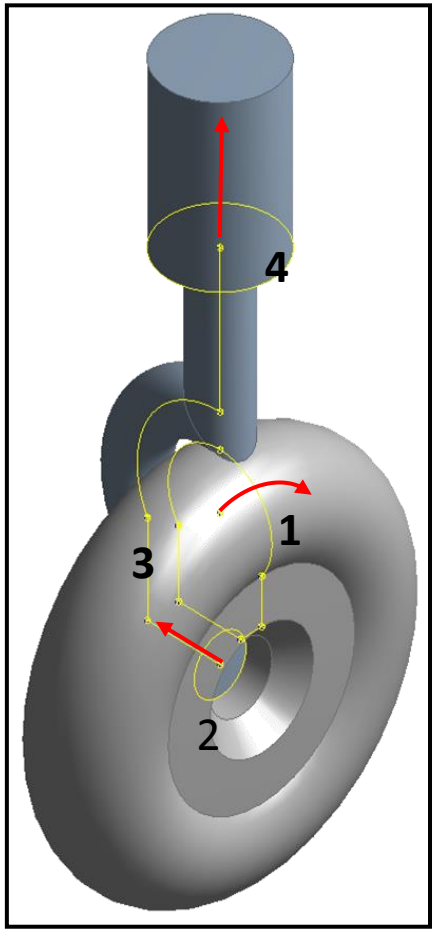
- In the Details View click in the box to the right of Path, select Sketch3 in the Outline and click Apply back in the Details View. Click Generate.



Preparation

Planning

- **Finally, we need to create the circular sketch (4) which will be used as the basis for an extrusion forming the large cylinder.**

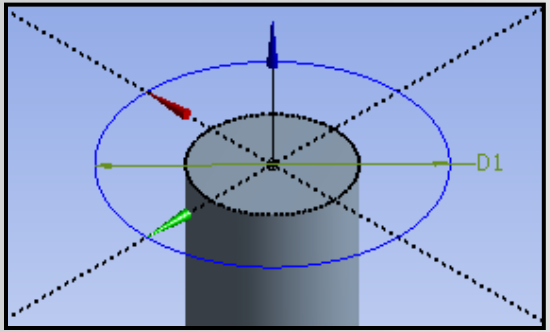


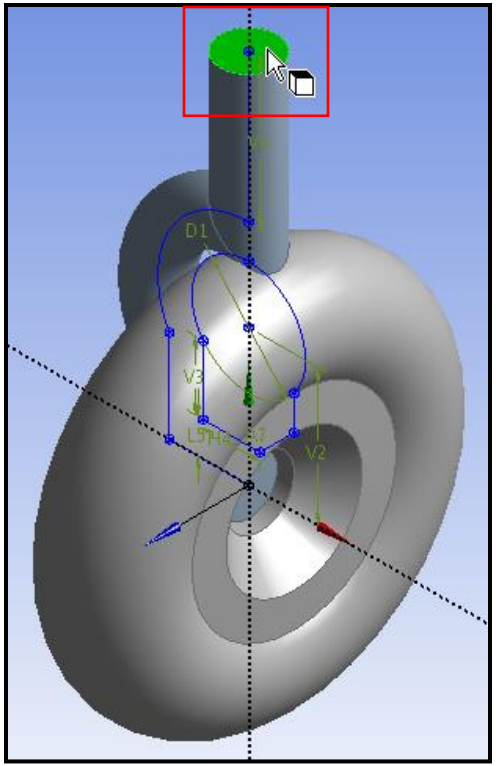
Sketching on a Face

Create a Sketch on a Face

- Select the Face Selection Filter.

- Select the top surface of the model as shown.
- Switch to Sketching Mode and, using the Circle Tool in the Draw Toolbox, create a circle centred on the origin of the new plane that has been created on the selected surface.
- Dimension the diameter to 10inch



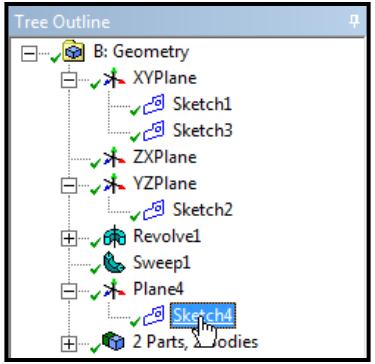


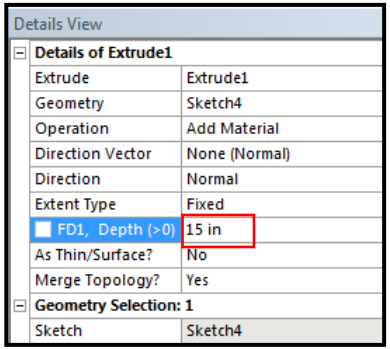
Extrude

Extrude the Sketch

- Switch to Modeling Mode.
- Notice that the action of switching to Sketch Mode with a selected face automatically created a plane and sketch on the selected face.
- Select Sketch4
- Select Extrude from the Create menu of main toolbar.

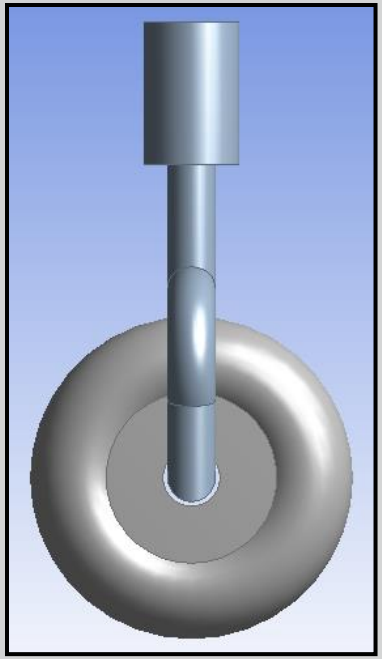
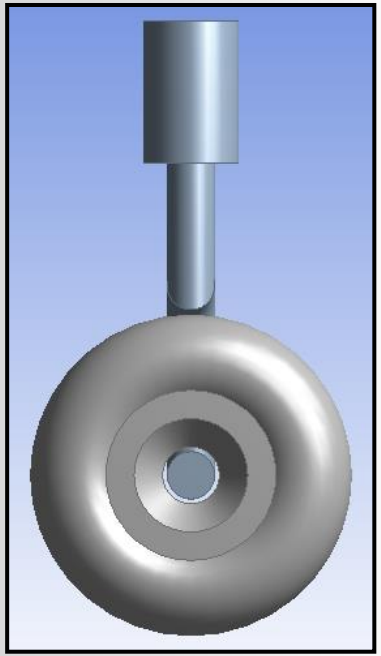
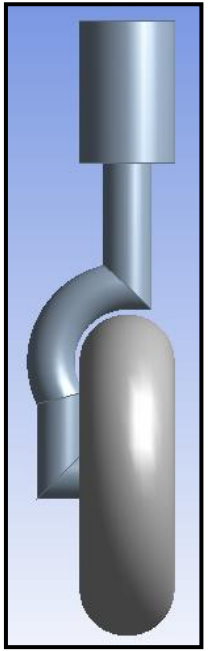
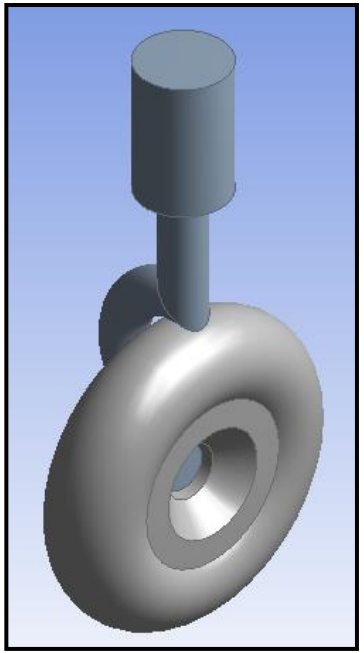
- Under the Details View set Depth to 15inch as shown.
- Generate.





Completion

This Completes the workshop.

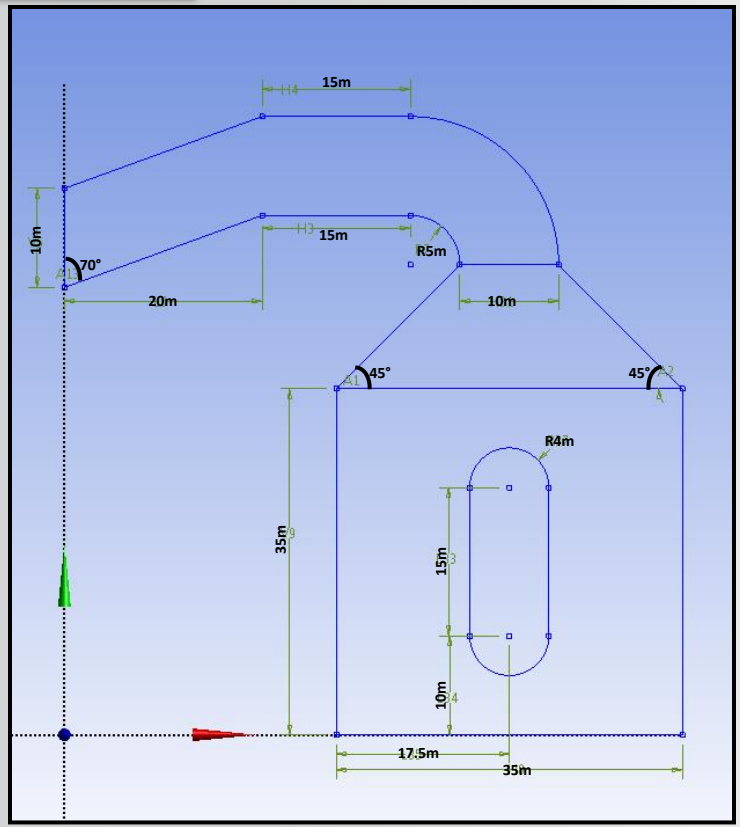





Saving the Project

- This completes the workshop.
- From the main menu select File → Close DesignModeler
  - Workbench will save any application data.

- From the Workbench Project Page use the file menu and save the project as “DMWS3.2.wbpj” to your working folder.

Optional Challenge



- All dimensions are in meters

Hint

- Draw sketch randomly without considering dimensions
- Apply constraint
- Apply dimensions