Practice #1b: Plane wheel

Objectives:

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

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.
**P1:Practice**

**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

**P1:Practice**

**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 millimeter
- Go to Units tab and select Inch.
  - Units → Millimetre

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 10mm in the Details View.
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.

1. Click the centre point of the circle.
2. Click the X Axis and position the dimension.

In the Details View:
- Set the new vertical dimension to 12 mm.
- 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:
  - D1: 10 mm
  - V2: 12 mm
- Edges: 1
  - Full Circle: 0

**P1: Practice**

**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.

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**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 2.5mm.
- 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

- 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

- 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.
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 6mm.

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 5mm.
**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 3mm in the Details View.

**Angle Dimension**

Add Dimensions
- Select the two edges in sequence as shown below.
- 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.
**P1:Practice**

**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 the Sketch (Continued)

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

- Click Generate.
Revolve

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

END of part 1

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 5mm 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 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)
**P1: Practice**

**Review**

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 13mm.
**P1: Practice**

**Sweep**

Create a Sketch (Continued)
- Switch to Modeling Mode using the tab.
- Select Sketch2 from the Outline.

- 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.

**P1: Practice**

**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.
**P1: Practice**

**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 10mm

**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.
  - Extrude

- Under the Details View set Depth to 15mm as shown.
- Generate.
P1: Practice
Completion

End...

END 😊