1 Project 2 - Flange Manifold Part

1.1 Instructions

This project focuses on additional sketching methods and sketching commands. Revolve and Work features are also introduced. The part being modeled is from machine tooling used in the manufacture of an automobile engine. This flange detail would have several probes attached to it that measure the fluid flow through the flange. The challenge is that the probes need to allow for maximum clearance away from the engine.

1: Create a new part using the Standard (mm).ipt template.
   ■ On the Quick Access toolbar, click New.
   ■ In the New File dialog box, click the Metric tab.
   ■ Select Standard (mm).ipt.
Click OK.

2: Create sketch centerline
- Start the Project Geometry Tool
- Sketch Tab | Draw Panel | Project Geometry
In the browser select the **Y Axis** to project a reference line into sketch.

Click **Done** on the Marking Menu

Select the projected reference line in the graphics window and then change it to a centerline by selecting the **Centerline** format from the ribbon. **Sketch Tab | Format Panel | Centerline**

3: Sketch basic shape

Start the Line tool.

In this project you will sketch the approximant profile of the part completely without dimensions, and then add the required geometric and dimensional constraints to complete the sketch.

Scrub over the projected origin point and drag to the right approximately 7 mm.

This will line up you cursor with the origin point.
Note: The term “Scrub” refers to passing the cursor over geometry in the graphics window without clicking on it in order to use that geometry in reference to another sketch element being created. Autodesk Inventor will try to infer what areas to reference, but sometimes you may have to Scrub over a point or other sketch element before it will be referenced.

A dotted line will show that you are lined up with the point.

- Left-Click to create the starting point of the first line segment. Drag to the right approximately 17 mm. Click to except this line segment.

- Drag up approximately 5 mm. Click to except this line segment.
■ Drag to the right approximately 15 mm. Click to except this line segment.

■ Drag up approximately 10 mm. Click to except this line segment.

■ Drag to the left approximately 10 mm. Click to except this line segment.
- Drag up approximately 15 mm. Click to except this line segment.

- Drag up and to the right to create an angled line segment similar to the one shown.
Drag up approximately 5 mm. Make sure the Parallel Constraint glyph is shown next to the cursor to insure this line segment is parallel to the other vertical line segments. Click to except this line segment.

Drag to the left approximately 15 mm. Click to except this line segment.
Drag up approximately 31 mm. Click to except this line segment.
Scrub the first starting point to line up this line segment with that point. When the dotted line is shown, and the line is parallel to the other horizontal lines (or perpendicular to the previous line segment) click to except.

Close the loop by picking the last line segment back to the first start point.
4: Constrain Sketch

- Place a **Coincident** Constraint between the bottom line segment and the projected Origin point.

- Place linear dimensional constraint for the overall part length. Enter 88 mm in the edit dimension field.

- Place an inner diameter dimensional constraint. Using the **General Dimension** tool from the **Sketch** tab on the ribbon, select the projected reference centerline and the inside edge of the part profile.
Notice that the dimension placed is a diameter. Enter **14.02 mm** in the edit dimension field.
Place the following diameter dimensional constraints.
Place the following linear dimensional constraints.
Place an angular dimensional constraint. Using the General Dimension tool from the Sketch tab on the ribbon, select the angled line segment and a vertical line segment. Enter 30 in the edit dimension field.

The sketch profile is now fully constrained.

Exit the sketch.

5: Revolve Base Feature.

Start the Revolve Tool

Model Tab | Create Panel | Revolve
Because there is only one closed loop exists in the sketch, the sketch profile is automatically selected.
Click OK

6: Using the Viewing tools reposition the view as shown.
7: Change Part Color

- On the Quick Access Toolbar pick **Zinc** from the part color drop down list.
8: Add Cavity Hole and End Tapped Hole
   - Start the Hole Tool.
     - Place a Concentric Hole on the top face with the following options:
       - Termination: Distance
       - Diameter: 20 mm
       - Depth: 44 mm
     
     Click OK
Start Hole Tool
Place a Concentric Hole on the top face with the following options:
Type: Tapped Hole
Thread Type: ANSI Metric M Profile
Size: 30
Designation: M30x2
Termination: Distance
Depth: 21 mm
Full Depth

Click OK
9: Add Extrusion Cut
   ■ Start a new sketch on the top surface

   ■ Start the Center Point Circle Tool
     Sketch Tab | Draw Panel | Circle

     Pick the projected center point for the Center Point Circle starting point.
     Drag away from the center. Enter 38 mm in the direct entry field. Press Tab to lock in the value.
     Press ENTER to except the circle.
Exit Sketch

Start the Extrude Tool
Pick both profiles inside the sketched circle.

Select the **Cut** option
Enter 2.3 mm in the entry field
Click **OK**
10: Add bolt pattern
- Create Sketch on bottom face shown.

- Start the **Line** Tool
  Draw vertical line starting from the center point **31.5 mm** long

- Exit the Sketch

- Start **Hole** Tool
  Place a **From Sketch** Hole picking the line endpoint in the sketch with following options:
  - Type: **Tapped Hole**
  - Thread Type: **ANSI Metric M Profile**
  - Size: **6**
  - Designation: **M6x1**
Termination: To
   Pick the top surface of the bottom flange area for the “To” termination surface
Full Depth

Click OK

- Start the Circular Pattern tool
  Model Tab | Pattern Panel | Circular
- Select the M6 tapped hole feature
  Select the outside diameter surface of the flange for the Rotation Axis

- Enter Pattern Placement: 6
  Angle: 360 deg

  Click OK
11: Create rotational location holes
   Create a new sketch on the bottom surface

- Change the sketch line format to construction by selecting the **Construction** format from the ribbon.
  
  **Sketch Tab | Format Panel | Construction**

  ![Construction Options]

- Start the **Center Point Circle** Tool
  
  Draw a circle **30 mm** in diameter

- Sketch two lines as shown.
  Starting from the center point and picking the construction circle for the second point
  
  **Note:** Make sure these lines are **NOT** perpendicular to each other.
Place a dimensional constraint between the line endpoints. Enter 19 mm into the edit dimension field.

Place a dimensional constraint between the upper line endpoint and the projected XY Plane.
- With the Edit Dimension field active, click on the 19 mm dimension. The dimension reference will be displayed in the Edit Dimension field (For this example it is shown as d55, it will be different for you)
- Create an equation by adding a forward slash “/” (for divide) and a “2”
- Click OK
- The value of this dimensional constraint will now be half the distance between the two line endpoints.
■ Exit the Sketch

■ Start Hole Tool
   Place a **From Sketch** hole picking both line endpoints in the sketch with following options:
   - **Style**: Counterbore
   - **Type**: Tapped Hole
   - **Termination**: Distance
   - **Thread Type**: **ANSI Metric M Profile**
   - **Size**: 4
   - **Designation**: M4x0.7
   - **Counterbore Diameter**: 8.02 mm
   - **Counterbore Depth**: 4 mm
   - **Overall Depth**: 18 mm

   Click **OK**
12: Reposition the view as shown
13: Create a Work Plane
   - Start the Work Plane tool
     Model Tab | Work Features Panel | Plane

   - In the browser click the Z Axis for the first reference
     Pick the YZ Plane for the second reference

   - Drag the direct manipulation arrow back until the work plane is shown at -30.00 deg
     Click OK

14: Create an Offset Work Plane
   - First you need to get a measurement for the offset.
     Click the model and click Edit Sketch to activate the first profile sketch you created.
Project 2 – Flange Manifold Part
- Draw a construction line parallel to the angled line in the sketch

- Start the Measure Distance tool
  **Tools Tab | Measure Panel | Distance**

  Click on the original angled line and the new construction line to get the measurement between them. A distance of 5.61 mm is shown in the **Measure Distance** dialog box.

  Copy this dimension for use in the next step.

  Exit the sketch.
Start the **Work Plane** tool
Click-Hold on the previous Work Plane and Drag away from it.
Enter **5.61 mm** into the direct entry field

Click **OK**
Right click on Work Plane1 and click **Visibility** on the Marking Menu
This will turn the visibility of this work plane OFF

Notice in the browser that all work features required to create *Work Plane2* are located as children under *Work Plane2*. They are shown as gray icons because their visibility has been turned OFF.
15: Create Manifold Hole Pattern On Round Angled Surface

- Create a new sketch on Work Plane2

- Start the Point tool
  Sketch Tab | Draw Panel | Point

Place a Point approximately -51 mm directly above the project Origin point
- Place a Vertical constraint between the drawn point and the projected Origin point.
- Place a dimensional constraint of 51 mm between the two points

- Exit Sketch

- Start Hole Tool
  Place a From Sketch hole picking both line endpoints in the sketch with following options:
  - Style: Countersink
  - Type: Taper Tapped Hole
  - Termination: To
  - Thread Type: NPT
  - Size: 1/4
  - Countersink Diameter: 23 mm
  - Countersink Depth: 2 mm
For “To” Termination hover cursor over the model until the Select Other drop down list appears. Select the inside diameter face.
- Click OK

- Start the Circular Pattern tool
  Select the 1/4 NPT tapped hole feature
  Select the outside diameter surface of the flange for the Rotation Axis
  Enter Pattern Placement: 6
  Angle: 360 deg
Click OK

16: Apply Chamfers

- Start the Chamfer tool
  
  \[ \text{Model Tab | Modify Panel | Chamfer} \]

- Select the top edge
  
Enter Distance: 1.5 mm into the Mini-Tool Bar

  Click Apply

- Change Chamfer style to Distance and Angle
  
  Select top surface adjacent to the tapped hole and top edge of the tapped hole
  
Enter Distance: 1.5 mm and Angle: 60 deg

  Click Apply
■ Change Chamfer style to **Distance**
Select top surface of bottom flange
Enter Distance: **2 mm**
Click **Apply**

■ Select the two bottom edges shown
Enter Distance: **0.5 mm**
Click **OK** to accept and exit the **Chamfer** tool
17: Apply Fillets

- Start the Fillet tool
  
  **Model Tab | Modify Panel | Fillet**

- Select edge as shown
  Enter Radius: **4 mm** into the Mini-Tool Bar
  Click **Apply**
Add Fillet Set
- Select edge as shown
  Enter Radius: 8 mm into the Mini-Tool Bar

- Click Add Constant Fillet Radius Set

- Select edge as shown
  Enter Radius: 3 mm into the Mini-Tool Bar

- Notice that both the 8mm and 3mm radiuses are created as one Fillet feature in the browser.
18: Save Part

- On the Quick Access toolbar, click Save.

- In the Save As dialog box, enter file name FlangeManifold.ipt

- Click Save