1. Project 3 - Air Cover Part

1.1 Project 3

This project focuses on advanced sketching methods. Shell and sweep features are also introduced.

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: In the active sketch, sketch the following full constrained profiles. Make sure you have 2 profiles in the one sketch as shown. Also note the position of the projected origin point; this will position the origin planes so that they can be used for creating features.
3: Create main part body
   - Start the **Extrude** tool
     Select the upper profile

   - Select the **Symmetric** direction. Drag the direct manipulation arrow, or enter **240 mm** into the Mini-Toolbar for the extrusion length.
     Click **OK**
In the browser expand *Extrusion1* and right-click on the consumed sketch. Select *Share Sketch* from the shortcut menu.

*Sketch1* is now available for use in more than one feature.
Start the **Extrude** tool
Select the lower profile

Select the **Symmetric** direction. Drag the direct manipulation arrow, or enter 200 mm into the Mini-Toolbar for the extrusion length.
Click **OK**
In the browser, right-click Sketch1. Select Visibility from the shortcut menu to turn off the sketch in the graphics window.

4: Change Part Color
On the Quick Access Toolbar pick Yellow (Dark) from the part color drop down list.
5: Add feature fillet
   - Start the **Fillet** tool
     - On the Mini-Toolbar pick the **Select Feature** option
Select the second feature you created (Extrusion2)

Enter a radius of 10 mm into the Mini-Toolbar
Notice that all edges of the feature have been selected

Click OK
6: Add additional fillets
   - Start the Fillet tool
     Select the edge between Extrusion1 and Extrusion2
     Notice that this edge is treated as a continuous loop because the fillet placed in the last step
     Enter 6 mm in the Mini-Toolbar

   Click Apply
Enter a radius of 30 mm into the Mini-Toolbar
Select the four corners of the upper feature (Extrusion1)
Notice that you can select edges that are not visible in the current view without changing the graphics window view by hovering over different parts of the model.

Click OK
7: Create Shell feature
   - Start the Shell tool
     Model Tab | Modify Panel | Shell

   - In the Shell Dialog Box enter 3 mm for the Thickness option
     Select the top surface of the part for Remove Faces

   Click OK

8: Add boss extrusion
   - Create a new sketch on the left side flat surface
   - Create a constrained circle as shown
9: Add Hole
   - Start the **Hole** tool
   - Create a **Concentric** hole on the previous extrusion
   - **50 mm** diameter
   - To inside face
10: Add bolt pattern
- Create a new sketch on the boss extrusion
  Sketch the point as shown

- Start the **Hole** tool
  Create a **From Sketch** tapped hole
  Type: **ANSI Metric M Profile**
  Size: 5
  Designation: **M5x0.8**
  Termination: **To**
Select the inside face for the To Termination
Click OK

- Start the Circular Pattern tool
  Select the M5 tapped hole feature
  Select the outside diameter of the boss extrusion
  Enter Pattern Placement: 4
  Angle: 360 deg

  Click OK
11: Create ribs on the model

- Start the **Work Plane** tool

- Click-hold the left outer surface of the part
■ Drag to the right -23.5 mm
  Click OK

■ Click OK

■ Project reference part edges into the sketch at the point where the sketch cuts through the part using the Project Cut Edges tool

■ Click F7 (Slice Graphics) to show sketch.

■ Sketch the following profile

Exit Sketch
Start the **Extrude** tool
- Select the sketch profile
- Enter 3mm in the direct entry field
- Select the **Direction 2** option
- Click **OK**
12: Pattern extruded rib feature
   - Start the **Rectangular Pattern** tool
   - Select the extruded rib feature
     - Select the top edge of the part for **Direction 1**
     - Column Count: 8
     - Column Spacing: 15 mm
   - Click **OK**

13: Create Sweep feature
   - Create a new Sketch
   - Click the **XY Plane** from the browser
- Project reference part edges into the sketch at the point where the sketch cuts through the part using the **Project Cut Edges** tool

- Click **F7 (Slice Graphics)** to show sketch.

- Sketch the following closed profile

- Exit the Sketch

- Create a new Sketch
  Select top surface
If top edges are not automatically projected into the sketch use the Project Geometry tool to project the top edges.

- Exit the Sketch

- Start the Sweep tool
  
  **Model tab | Create panel | Sweep**

- Select the sketched profile from the first sketch for the Profile
Select the projected edge from the second sketch for the Path

Click OK

14: Save Part
- On the Quick Access toolbar, click Save.

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