Sports drink bottle tutorial
Pro|ENGINEER Wildfire 3.0
Schools & Schools Advance Edition
Introduction

The Sports Drink bottle tutorial introduces Students to the skills and techniques needed to develop design ideas within Pro/ENGINEER Wildfire 3.0.

During these tutorials Users will learn how to create parts and assemblies within Pro/ENGINEER Wildfire 3.0, and explore the D&T activities that are involved in the product design process.

This Tutorial and Teacher Resource has been produced by PTC® and in support of the PTC Design & Technology in Schools programme.

Pre-requisites

Pro/ENGINEER Wildfire 3.0 Schools Edition
or
Pro/ENGINEER Wildfire 3.0 Schools Advanced Edition
or
Pro/ENGINEER Wildfire 3.0 University Plus Edition
or
Pro/ENGINEER Wildfire 3.0 Student Edition

This tutorial contains screen and menu images taken from the Schools Edition so Users of other Pro/ENGINEER Editions may notice some slight differences.

This tutorial has also been based on the use of Pro/ENGINEER start parts & templates supplied as part of the PTC D&T programme. While this tutorial can be used with other Pro/ENGINEER start parts there may be changes required in terms of view orientation, datum plane and coordinate system references etc.

This tutorial requires no previous modelling experience in Pro/ENGINEER; however the User should be able to navigate the Pro/ENGINEER Wildfire User Interface.

Pro/ENGINEER Wildfire requires the use of a 3 button mouse. If possible a mouse with a combined middle wheel & button can improve User interaction with Pro/ENGINEER Wildfire.
Abbreviations and terminology used within this tutorial

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The aim of the tutorial is to introduce students to the basic solid-modelling process and techniques used within Pro|ENGINEER Wildfire 3.0.

Installation and setup

These Installation notes have been compiled based on a directory structure used as part of the PTC D&T programme, the UK CAD in Schools initiative and the deployment of Pro|ENGINEER. Users not part of this programme can still use this tutorial but may need to adapt either their Pro|ENGINEER configuration files or the directory structure used in the tutorial.

Please ensure you have the required materials (LDPE..mat or LDPE.mtl) in the material library folder within pro_standards.

Have the students create a directory under their documents or network folder called “bottle”.
Pro|ENGINEER functionality addressed in this tutorial.

- Sketching
  - 2D geometry creation & modification
    - Circles, Lines, Centrelines, Trimming.
    - References
  - Geometric & dimensional constraints
    - Weak, Strong & Locked dimensions
    - Linear and angular dimensional constraints
    - Equal radius and tangent geometric constraints

- Modelling
  - Revolve
  - Shell
  - Extrude
  - Patterning
  - Parametric modifications

- Assemblies
  - Assembly constraints

ICT areas addressed in this tutorial

- Modelling
- Communication

D&T subject areas addressed in this tutorial

- CAD
  - Parametric feature based solid-modelling
  - Assemblies

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Background

It is widely accepted that when taking physical exercise or training it is critical to remain properly hydrated. When working out in a gym a source of clean fresh drinking water is almost certainly close at hand. However one of the most popular forms of exercise is jogging or running; due to the nature of this type of exercise the runner typically has to carry a bottle of water or energy drink.

While there are many different bottled water and sports drink products on the market, many come in a basic bottle design which, when running, can be difficult or uncomfortable to carry on long distances.

This tutorial will address the market need for an ergonomic drinks bottle suitable for use by runners. The bottle should be designed to hold 500 millilitres and be easy and comfortable to carry while running.
Lesson one – Bottle production methods

Aim:
You will learn about the different bottle production methods concentrating on thermo plastic polymer materials and blow moulding. You will need to combine this understanding with early capability with Pro|ENGINEER when designing and detailing your own bottle design later in the module.

Learning objectives:
By the end of this lesson you should:

° Be aware of the production methods used to create plastic bottles.
° Understand the limitations blow moulding techniques impose on bottle design.
° Be able to suggest bottle shapes that might be manufactured

The main part of the lesson teaches you about plastic bottle production methods including the materials used. The focus will be on the manufacturing process/materials and the limitations these impose on the shape.

Your teacher will introduce the module including the challenge making links to health and well-being. You should understand the connections with sport and food technology where you may be designing the drink to go in the bottle making sure it is healthy, nutritious and provides an energy boost.

Homework
Research existing bottle designs for sports drinks. Draw annotated pictorial sketches of at least three. Bring examples in for next lesson
Lesson two – Product analysis

Aim:
During this lesson you will have the opportunity to develop a clear understanding of existing sports drink bottles, how they function, their shape, method of manufacture and the information displayed on the outside.

Learning objectives:
By the end of the lesson you should:

° Be aware of the different shapes of bottle and how this can be linked to a specific company.
° Understand how the shape of plastic bottles is limited by the blow moulding manufacturing process.
° Be able to suggest shapes that might appeal to the user and can be manufactured.

This lesson engages you in product analysis techniques to help you gain a clear understanding of the range of sports drinks bottles, their materials, design, closure, holding, etc.

Useful resources to help with aspects of design can be found at:
http://www.standards.dfes.gov.uk/keystage3/respub/design/foreword/

Homework
Generate design ideas based on understanding of Pro/ENGINEER solid creation from extrude and revolve
Lesson three – Modelling the bottle

Aim:
In this lesson students will learn about the Pro|ENGINEER graphical user interface (GUI) and be taught how to create the body of a sports drink bottle.

Learning objectives:
By the end of this lesson students should:

° Be aware of the part creation techniques using Pro|ENGINEER.
° Understand the principles of sketch based features and direct features.
° Be able to create valid sketch geometry.
° Be able to create solid shapes using extrude and revolve features.
° Be able to shell the shape to create a hollow part.

Task 1: Set Working Directory

1. Start Pro|ENGINEER Wildfire

2. In the Navigator Window (down the left-hand side of Pro|ENGINEER), browse to the “bottle” folder.

   If the Navigator is not displaying Folders left-click the Folder tab at the top Navigator Window.

3. Right-click the “bottle” folder, and in the menu that appears select Set Working Directory.
Task 2: Creating a new Pro/ENGINEER part

4. From the Pro/ENGINEER top toolbar left-click Create New File. In the dialog box that appears enter “bottle”.

Notice that Part is selected as the default Type

5. Left-click OK to accept the settings and create the new Pro/ENGINEER part file.

When the part opens you should see the default Datum Planes, FRONT, TOP, & RIGHT, and the default coordinate system DEFAULT_CSYS displayed in the graphics window and Feature Navigation pane.

For the purposes of this activity the DEFAULT_CSYS is not required;

6. From the Pro/ENGINEER top toolbar left-click Coordinate System on/off to turn off the display.
Task 3: Creating the basic bottle feature

The basic bottle shape will be created with an Extrude Feature.

7. From the Feature toolbar (down the right-hand side of Pro|ENGINEER) select Extrude Feature.

Pro|ENGINEER will display the Extrude Feature tools and options in the ‘Dashboard’ along the bottom of the Pro|ENGINEER window.

An Extrude Feature is a Sketch Based Feature; the next step is to define the sketch.

8. In the Extrude Feature Dashboard left-click the word Placement, this will open a small Dashboard slide-up panel, select Define.

Before you start sketching, Pro|ENGINEER needs to know where to place the Sketch and how it is to be oriented. Pro|ENGINEER will issue a prompt along the bottom of the Pro|ENGINEER window asking you to;

Select a plane or surface to define sketch plane.

Pro|ENGINEER will also display the Sketch dialog which captures the selection of Sketch Plane and Sketch Orientation information.
9. In the Pro|ENGINEER Graphics Window move the cursor over the TOP datum plane and select it with a left-click. This will populate the Plane data box.

Pro|ENGINEER will then automatically suggest/select the FRONT datum plane as the Reference Plane to define the Sketch Orientation and Sketch view direction.

10. To accept these references and enter the Sketcher select Sketch

Once in the Sketcher, Pro|ENGINEER will orient the view to look directly onto the Sketch Plane. If this doesn’t happen, in the top toolbar select to reorient the view.

At this point you no longer need to see the Datum Planes.

11. In the top toolbar select to turn off the display of Datum Planes.

Based on the selection of the TOP datum Plane as the sketch plane and the FRONT datum plane as the orientation plane, Pro|ENGINEER has automatically created two reference lines in the sketch from the two Datum Planes. The reference lines will be used to position the sketch geometry for the Extrude Feature.

Once in the Sketcher, Pro|ENGINEER will display the Sketcher toolbar down the right-hand side of the Pro|ENGINEER Window.

12. From the Sketcher toolbar select Create Circle
13. Move the cursor over the intersection of the two reference lines. Pro|ENGINEER will recognise the intersection as an ‘attraction point’. Left-click to select this intersection as the start point/centre of the circle ($X_1$).

14. Move the cursor away from the centre and Pro|ENGINEER will sketch a circle; to create the circle left-click once more ($X_2$). Don’t worry about the diameter/radius value of the circle at this point.

15. The next circle needs to be smaller in diameter and positioned along the horizontal reference line. Move the cursor over the horizontal reference line, again Pro|ENGINEER will recognise the line as an ‘attraction point’, left-click to locate the centre ($X_1$).

16. Again, move the cursor away from the centre and Pro|ENGINEER will sketch a circle; to create the circle left-click once more ($X_2$).

17. To exit create Circle press the middle mouse button once, (middle-click).

Pro|ENGINEER will automatically create dimensions to fully constrain the geometry. These dimensions are typically grey in colour denoting they are “weak” dimensions.

**Note:**
There are 3 types of sketch dimension;
• **Locked** – the dimension is locked to its value. This value cannot be modified either directly or indirectly. The dimension has to be un-locked before its value can be modified.

• **Strong** – the dimension can be modified but only directly by the user

• **Weak** - the dimension can be modified directly (by explicitly changing the value) or indirectly (by changing other surrounding dimensions/geometry).

The next step is to add tangential lines to create the bottle profile.

18. From the Sketch toolbar left-click the small upturned arrow next to **Create Line** and from the line pull-out menu select **Create Tangent Line**.

19. Move the cursor over the first circle and left-click to start the line(\(X_1\)). Move the cursor to the smaller circle and left-click to create the tangent line(\(X_2\)).

20. Left-click **Create Tangent Line** and sketch another tangent line at the bottom of the two circles.

To complete the profile the inner sections of the two circles need to be removed.

21. From the Sketch toolbar select **Dynamic Trim** and select the circles at the points indicated in the illustration.
The Sketch profile now needs to be dimensioned and the required dimension values entered.

22. From the Sketch toolbar select **Create Dimension**; left-click the vertical reference line and then the centre of the smaller circle; to place the dimension move the cursor above the profile and middle-click.

23. Next double-left-click one of the circles, move the cursor outside the circle and place the Diameter dimension with a middle-click. Repeat this for the other circle.

*To create a radial dimension select the circle with a single left-click. To create a Diameter dimension select the circle with a double-left-click*

*Notice, these new, user defined, dimensional constraints have been created in a different colour than the automatically created dimensions, these newly created dimensions are “Strong” dimension.*

The sketch now has the required dimensioning scheme, but the values are still incorrect.
24. From the Sketch toolbar left-click **Select Item** and double-left-click the diameter dimension of the smaller circle, enter a value of **25mm** (hit the return key to enter the value). Repeat this process and change the diameter of the larger circle to **65mm** and the horizontal dimension between the circles to **65mm**.

Note: Based on configuration settings used for the PTC CAD in Schools programme any modified dimension is converted into a “Locked” dimension.

(Pro|ENGINEER config.pro setting “sketcher_lock_modified_dms = yes”)

To help with any future drawings an Axis Point will be added to the centre of each circle. These points will create axes as part of the extrusion.

25. From the top toolbar select **Sketch**, and from the pull down menu select **Axis Point**. Move the cursor to the centre of each circle and left-click to create the Axis point.

26. The Sketch is now complete. To accept and exit the Sketcher, left-click **Accept Sketch ✓**.

27. The view will still be in the same orientation used for sketching. From the top toolbar select **Saved View List** and from the pull-down menu select either
**ISOMETRIC** or **TRIMETRIC** to return to a more suitable view orientation.
Pro|ENGINEER will automatically create an Extrude Feature with a default/arbitrary depth.

28. The depth value can be changed dynamically; move the cursor over the small white square (the drag handle) and left-click-drag. Drag the extrusion to a depth of **160mm**.

Alternatively the extrude depth can be explicitly specified in the Dashboard.

29. The yellow arrow indicates the direction of the extrude, in this case the extrude needs to be below the TOP Datum Plane; to change the extrude direction left-click the yellow arrow in the graphics window or select **Flip Direction** in the Dashboard.

30. To complete the Extrude feature select **Complete Feature** at the far right-hand side of the Dashboard.

31. At this point save the part. From the top toolbar select **Save File** and in the **Save** dialog select **OK**.

**Task 4: Creating the bottle handle**
The handle will be created by removing material from the basic bottle solid.

32. From the Feature toolbar (down the right-hand side of Pro|ENGINEER) select Revolve Feature.

Pro|ENGINEER will display the Revolve Feature tools and options in the ‘Dashboard’ along the bottom of the Pro|ENGINEER window.

A Revolve Feature is a Sketch Based Feature; the next step is to define the sketch.

33. In the Revolve Feature Dashboard left-click the word Placement, this will open a small Dashboard dialog, select Define.

Before you start sketching Pro|ENGINEER needs to know where to place the Sketch and how it is to be oriented.

Pro|ENGINEER will issue a prompt along the bottom of the Pro|ENGINEER window asking you to;

Select a plane or surface to define sketch plane.

Pro|ENGINEER will also display the Sketch dialog which captures the selection of Sketch Plane and Sketch Orientation information.
34. In the Pro|ENGINEER Feature Navigator move the cursor over the **FRONT** datum plane and select it with a left-click. This will populate the **Plane** data box.

Pro|ENGINEER will then automatically suggest/select the **TOP** datum plane as the **Reference** Plane to define the Sketch Orientation and Sketch view direction.

35. To accept these references and enter the Sketcher select Sketch

Once in the Sketcher Pro|ENGINEER will orient the view to look directly onto the Sketch Plane. If this doesn’t happen, in the top toolbar select \( \square \) to reorient the view.

Based on the selection of the **FRONT** datum Plane as the sketch plane and the **TOP** datum plane as the orientation plane, Pro|ENGINEER has automatically created two reference lines in the sketch.

36. From the Sketcher toolbar select Circle \( \bigcirc \) and using the same process used previously position the cursor where the centre of the circle is required, left-click to select this location as the start point/centre of the circle \( (X_1) \).

37. Move the cursor away from the centre and Pro|ENGINEER will sketch a circle; to create the circle left-click once more \( (X_2) \).

Don’t worry about the diameter/radius value of the circle at this point.

38. Now move the cursor down below the first circle and try to position the cursor so it is almost directly below the centre of the first circle.

As you do this you will notice Pro|ENGINEER automatically detects an imaginary vertical line to help you locate the centre of the new circle vertically aligned beneath the centre of the first circle.

Pro|ENGINEER indicates this geometric constraint by displaying a small red dash at the centre of the first circle and at the cursor point. Left-click to select this location as the start point/centre of this circle \( (X_3) \).
39. Move the cursor away from the centre and Pro|ENGINEER will sketch a circle. This second circle needs to be created the same diameter as the first. As you move the cursor to increase the diameter of the circle Pro|ENGINEER will display “R1” next to both circles indicating an equal radius constraint exists between the two circles; left-click once more (X4) to create this second circle.

40. To exit create Circle press the middle mouse button once, (middle-click).

The next step is to join these circles to create the handle profile.

41. From the Sketch toolbar left-click the small upturned arrow next to Create Line and from the line pull-out menu select Create Tangent Line.

42. Left-click one side of the top circle followed by the corresponding side on the bottom circle. Repeat this process to create the tangent line on the other side of the circle. Middle-click to exit line creation.

To complete the profile the inner sections of the two circles needs to be removed.

43. From the Sketch toolbar select Dynamic Trim and select the circles at the points indicated in the illustration. Middle-click to exit Dynamic Trim.

A revolve feature requires an axis of revolution.

44. From the Sketch toolbar left-click the small upturned arrow next to Create Line and from the pull-out menu select Create Centerline.
45. Position the cursor along the horizontal reference line and left-click to place the first point of the centreline ($X_1$). Move the cursor down till the centreline snaps to vertical and a “V” is displayed to indicate a vertical geometric constraint will be created, then left-click to finalise creation of the centreline ($X_2$).

The next step is to dimension the handle profile to the required dimensioning scheme.

46. From the Sketch toolbar select Create Dimension.

47. Left-click the two sides of the handle profile ($X_1$ & $X_2$), and place the dimension with a middle-click ($X_3$).

48. The next dimension will position the handle horizontally: Left-click the left-hand vertical edge of the bottle ($X_1$), Pro|ENGINEER will recognise the edge and create an equivalent reference line. Then left-click the side of the handle profile ($X_2$), and then place the dimension with a middle-click ($X_3$).
49. To dimension the location of the centreline left-click the centreline and the edge of the bottle \((X_1 \text{ & } X_2)\), and place the dimension with a middle-click \((X_3)\).

50. The next dimension will define the distance of the handle from the top of the bottle. Left-click the top of the upper circle \((X_1)\) followed by the horizontal reference line \((X_2)\), and then place the dimension with a middle-click \((X_3)\).

51. Repeat this process for the bottom of the handle.

The next step is to change the dimensions to the required values.

52. From the Sketch toolbar left-click Select Item and double-left-click each newly created dimension and change their values to the following, (hit the return key to enter each value):

a. The width of the handle profile = 30mm  
b. Distance from handle to bottle side = 25mm  
c. Distance from bottle side to centreline = 30mm  
d. Distance from top of bottle to handle = 20mm
e. Distance from bottom of bottle to handle = 20mm

53. The sketch is now complete. In the Sketcher toolbar left-click **Accept Sketch** ✔ to accept and exit the Sketcher.

54. The view will still be in the same orientation used for sketching. From the top toolbar select **Saved View List** ▽ and from the pull-down menu select either **ISOMETRIC** or **TRIMETRIC** to return to a more suitable view orientation.

55. Pro|ENGINEER will automatically preview the Revolve Feature.

56. The default action is to “add” material but in this case the Revolve Feature needs to subtract/remove material. In the Revolve Feature Dashboard select **Remove Material**  

57. To complete the Revolve feature select **Complete Feature** ✔ at the far right-hand side of the Dashboard.

58. At this point save the part. From the top toolbar select **Save File** and in the Save dialog select **OK**.

**Task 5: Adding the bottle top**

The bottle is required to fit a standard bottle push-fit cap.

59. From the Feature toolbar left-click **Extrude Feature** .

Pro|ENGINEER will display the Extrude Feature tools and options in the ‘Dashboard’ along the bottom of the Pro|ENGINEER window.

An Extrude Feature is a Sketch Based Feature; the next step is to define the sketch.
60. In the Extrude Feature Dashboard left-click the word **Placement**, this will open a small Dashboard dialog, select **Define**.

Before you start sketching Pro|ENGINEER needs to know where to place the sketch and how it is to be oriented. Pro|ENGINEER will issue a prompt along the bottom of the Pro|ENGINEER window asking you to; ✗ Select a plane or surface to define sketch plane.

Pro|ENGINEER will also display the Sketch dialog which captures the selection of **Sketch Plane** and **Sketch Orientation** information.

61. In the Pro|ENGINEER Graphics Window move the cursor over the top face of the bottle select it with a left-click. This will populate the **Plane** data box.

Pro|ENGINEER will then automatically suggest/select the **FRONT** datum plane as the **Reference** Plane to define the Sketch Orientation and Sketch view direction.

62. To accept these references and enter the Sketcher select ![Sketch button](image).

Once in the Sketcher Pro|ENGINEER will orient the view to look directly onto the Sketch Plane.

If this doesn’t happen, in the top toolbar select ![Reorient view](image) to reorient the view.

63. From the Sketch toolbar select **Create Circle** ![Create Circle](image)
64. Position the cursor at the intersection of the two default reference lines and left-click \( (X_1) \) to position the centre of the circle.

65. Move the cursor away from the centre and left-click \( (X_2) \) to create the circle.

66. From the Sketch toolbar select **Create Dimension** \( \text{Ctrl+D} \). Double-left-click the circle and middle-click to place the Diameter dimension.

67. From the Sketch toolbar left-click **Select Item** \( \text{Ctrl} \) and double-left-click each newly created dimension and change the value to **50mm**.

68. The Sketch is now complete. In the Sketcher toolbar left-click **Accept Sketch** \( \text{Ctrl+P} \) to accept and exit the Sketcher.

69. The view will still be in the same orientation used for sketching. From the top toolbar select **Saved View List** \( \text{Ctrl} \) and from the pull-down menu select either **ISOMETRIC** or **TRIMETRIC** to return to a more suitable view orientation.

Pro|ENGINEER will automatically preview the Extrude Feature.

70. Double-left-click the extrude distance value and change it to **15mm**.

71. To complete the Extrude feature select **Complete Feature** \( \text{Ctrl+P} \) at the far right-hand side of the Dashboard.

At this point save the part.

72. From the top toolbar select **Save File** \( \text{Ctrl+S} \) and in the Save dialog select **OK**.
Task 6: Adding detail to the bottle – rounding the edges

The edges of the bottle need to be rounded.

73. From the Feature toolbar select **Round Feature**.

74. Select the two edges shown in the illustration; Left-click the first edge and then hold down the **Ctrl** key to add the second handle edge to the selection.

75. Pro|ENGINEER will preview a round of a default/arbitrary radius. The radius required for the handle is **8mm**, either drag the small white drag-handles until the value reaches 8mm or enter 8mm in the Round feature dashboard.

76. To accept/create this Round feature left-click ✔️ in the far right of the dashboard.

The next Round feature will round off the top and bottom of the bottle.

77. From the Feature toolbar select **Round Feature**.

78. Select the two edges shown in the illustration; Left-click the first edge and then hold down the **Ctrl** key to add the second handle edge to the selection.

79. Pro|ENGINEER will once again preview the round, this time it will suggest the previously used radius, however in this case the required round radius is **3mm**. Either use the drag-handles or enter 3mm in the dashboard.

80. To accept/create this Round feature left-click ✔️ in the far right of the dashboard.
The final Round feature is the base of the bottle neck.

81. From the Feature toolbar select **Round Feature**.
82. Select the edge shown in the illustration.
83. Pro|ENGINEER will again preview the round, this time it will suggest the previously used radius, in this case the required round radius is **3mm**. Either use the drag-handles or enter 3mm in the dashboard.

84. To accept/create this Round feature left-click in the far right of the dashboard.
85. At this point save the part. From the top toolbar select **Save File** and in the Save dialog select **OK**.
Lesson four – Computer Aided Manufacture (CAM)

Focus: Comparison of CNC and RP

Aim:
In this lesson you will be introduced to computer controlled manufacturing processes including high speed machining and rapid prototyping. Ideally by visiting a modern high-tech company or through video/web based resources.

Learning objectives:
By the end of this lesson you should:

° Be aware how 3D computer models can be manufactured by adding or subtracting material.
° Be aware of applications for CNC and RP and the role they can play in shortening the time from design to manufacture
° Understand the way CNC machining works and the limitations the process has on the shapes that can be made.
° Understand the different rapid prototyping methods.
° Be able to suggest shapes that can be made using CNC/RP technologies.

This lesson aims to give you a good understanding of modern Computer Aided Manufacture and is best done by visiting a commercial site to see both CNC machining and rapid prototyping. Bureaus are increasingly common often located at technology parks linked to universities and colleges.

A PowerPoint presentation is available that explains the difference between CNC and RP.

In addition, there are lots of useful resources on the web including:

Useful resources for this activity include:

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Lesson five – Complete bottle & start cap

Aim:
In this lesson students build on the experience of revolving the handle void and are taught how to revolve the cap with patterned ridges around the edge and a ‘snap fit’ lip inside.

Learning objectives:
By the end of this lesson students should:
° Be aware solids can be created using the revolve feature and features can be patterned
° Understand the concepts underlying solid creation using sketch based features.
° Understand that solids can be modified using direct features.
° Be able to create solids using revolve features
° Be able to modify solids using direct features

Task 7: Hollowing out the bottle
At the moment the bottle is solid, to make it hollow the inside of the bottle needs to be removed. In Pro/ENGINEER this hollowing-out is called a Shell feature.

1. From the Feature toolbar select **Shell**

As nothing is selected Pro/ENGINEER will apply the Shell to the entire bottle; that is, remove material from the inside of the bottle. However the bottle needs to be open at the top.

2. Left-click the top of the bottle to indicate this is ‘open’ end of the bottle.

3. Pro/ENGINEER will preview a default offset distance for the Shell feature, with either the drag handles or in the feature dashboard enter a Shell thickness of **1.5mm**

4. To accept/create this Shell feature left-click ✔ in the far right of the dashboard.

5. At this point save the part. From the top toolbar select **Save File** and in the Save dialog select **OK**.
Task 8: Adding the bottle top lip – push fit

The last feature to add to the bottle is a small lip around the bottle top for the push-fit bottle cap.

The Lip will be created by revolving a semi-circle around the bottle top to add material.

6. From the Feature toolbar (down the right-hand side of Pro|ENGINEER) select Revolve Feature.

Pro|ENGINEER will display the Revolve Feature tools and options in the ‘Dashboard’ along the bottom of the Pro|ENGINEER window.

A Revolve Feature is a Sketch Based Feature; the next step is to define the sketch.

7. In the Revolve Feature Dashboard left-click the word Placement, this will open a small Dashboard dialog, select Define.

Before you start sketching, Pro|ENGINEER needs to know where to place the Sketch and how it is to be oriented. Pro|ENGINEER will issue a prompt along the bottom of the Pro|ENGINEER window asking you to; Select a plane or surface to define sketch plane.

Pro|ENGINEER will also display the Sketch dialog which captures the selection of Sketch Plane and Sketch Orientation information.

8. In the Pro|ENGINEER Feature Navigator move the cursor over the FRONT datum plane and select it with a left-click. This will populate the Plane data box.

Pro|ENGINEER will then automatically suggest/select the TOP datum plane as the Reference Plane to define the Sketch Orientation and Sketch view direction.
To accept these references and enter the Sketcher select Sketch.

Once in the Sketcher Pro|ENGINEER will orient the view to look directly onto the Sketch Plane. If this doesn’t happen, in the top toolbar select to reorient the view.

Based on the selection of the FRONT datum Plane as the sketch plane and the TOP datum plane as the orientation plane, Pro|ENGINEER has automatically created two reference lines in the sketch. A further reference line is needed;

9. From the Pro|ENGINEER top toolbar select Sketch and in the pull-down menu that appears select References…

10. Select the top and outer edges of the bottle top extrusion, and in the References menu select Close.

11. Zoom into the bottle top. To do this position the cursor over the newly created reference line, hold down the Ctrl key and middle-drag downwards until the bottle neck is big enough to allow you to create the small lip. Alternatively, if you have a mouse with a middle wheel, scroll the mouse wheel with cursor centred over the middle of the neck to perform the zoom.
12. From the Sketch toolbar select **Create Circle**.

13. Position the cursor on the small vertical reference line and left-click to place the centre of the circle; move the cursor onto the horizontal reference line (near to where it intersects with the small vertical reference line), Pro|ENGINEER will display a small **T** denoting a Tangent constraint will be made, left-click to create the circle with this tangent constraint.

14. From the Sketch toolbar select **Dynamic Trim** and select the circle at the point indicated in the illustration. Middle-click to exit Dynamic Trim. This will leave just one half of the circle.

15. From the Sketch toolbar select **Create Line**. You may need to select the small up-turned arrow next to the **Tangent Line** and select **Create Line**.

16. Position the cursor at the bottom end of the semi-circle and left-click (X1) to start the Line, move the cursor to the other end of the semi-circle and left-click (X2) to connect the line. Middle-click to exit line creation.

As this profile is to be revolved the next step is to create an axis of revolution.

17. From the Sketch toolbar select the small up-turned arrow next to Create Line and from the pull-out menu select **Create Centreline**.
18. Position the cursor over the long vertical reference line and create the centreline. Middle-click to exit centre line creation.

19. From the Sketch toolbar left-click Select Items. Double left-click the dimension for the semi-circle radius and change it’s value to **1.5mm**.

20. The Sketch is now complete. In the Sketcher toolbar left-click **Accept Sketch** to accept and exit the Sketcher.

21. The view will still be in the same orientation used for sketching. From the top toolbar select Saved View List and from the pull-down menu select either ISOMETRIC or TRIMETRIC to return to a more suitable view orientation.

22. To complete the Revolve feature select **Complete Feature** at the far right-hand side of the Dashboard.
23. At this point save the part. From the top toolbar select **Save File** and in the **Save** dialog select **OK**.

**Task 9: Assigning material properties and other parameters**

While the bottle ‘model’ is finished, in that it’s geometric form is complete, Pro\|ENGINEER can capture more information which is an important part of the design and engineering process. This additional information includes material and project information.

24. From the Pro\|ENGINEER top toolbar left-click **Edit**, and from the pull-down menu select **Setup**...

25. This will open up the Pro\|ENGINEER menu-manager. Left-click **Material**.
26. Pro|ENGINEER will now display the Material dialog. Select LDPE (Low Density Polyethylene), left-click the assign button followed by OK. 

Pro|ENGINEER will now apply this material to the bottle solid model.

27. To close the menu-manager select Done.

The next step is to assign none geometric parameters to the bottle such as Description and Project.

28. From the Pro|ENGINEER top toolbar left-click Tools and from the pull-down menu select Parameters...

29. In the Parameter dialog fill in the Values for DESCRIPTION, MODELLED_BY and PROJECT. Click OK to accept your Parameters.
The bottle is now finished. At this point save the part.

30. From the top toolbar select **Save File** and in the Save dialog select **OK**.
Task 10: Creating the cap part in Pro|ENGINEER

31. From the Pro|ENGINEER top toolbar left-click Create New File. In the dialog box that appears enter “cap”.

Notice that Part is selected as the default Type

32. Left-click OK to accept the settings and create the new Pro|ENGINEER part file.

When the part opens you should see the default Datum Planes, FRONT, TOP, & RIGHT, and the default coordinate system DEFAULT_CSYS displayed in the graphics window and Feature Navigation pane.

For the purposes of this activity the DEFAULT_CSYS is not required;

33. From the Pro|ENGINEER top toolbar left-click Coordinate System on/off to turn off the display.

Creating basic cap feature

The cap will be created from a Revolve Feature. The creation of the cap Revolve Feature will use an ‘open’ profile; the sketch will be made up of a few straight lines and the ‘Feature’ will add thickness to these lines to create the solid model.

34. From the Feature toolbar (down the right-hand side of Pro|ENGINEER) select Revolve Feature.

Pro|ENGINEER will display the Revolve Feature tools and options in the ‘Dashboard’ along the bottom of the Pro|ENGINEER window.

*IMPORTANT: In the Dashboard select Thicken Sketch.

A Revolve Feature is a Sketch Based Feature; the next step is to define the sketch.
35. In the Revolve Feature Dashboard left-click the word **Placement**, this will open a small Dashboard dialog, select **Define**.

Before you start sketching Pro|ENGINEER needs to know where to place the Sketch and how it is to be oriented. Pro|ENGINEER will issue a prompt along the bottom of the Pro|ENGINEER window asking you to; Select a plane or surface to define sketch plane.

Pro|ENGINEER will also display the Sketch dialog which captures the selection of **Sketch Plane** and **Sketch Orientation** information.

36. In the Pro|ENGINEER Feature Navigator move the cursor over the **FRONT** datum plane and select it with a left-click. This will populate the **Plane** data box.

Pro|ENGINEER will then automatically suggest/select the **TOP** datum plane as the **Reference** Plane to define the Sketch Orientation and Sketch view direction.

37. To accept these references and enter the Sketcher select **Sketch**.

Once in the Sketcher Pro|ENGINEER will orient the view to look directly onto the Sketch Plane. If this doesn’t happen, in the top toolbar select ![reorient](image) to reorient the view.

Based on the selection of the **FRONT** datum Plane as the sketch plane and the **TOP** datum plane as the orientation plane, Pro|ENGINEER has automatically created two reference lines in the sketch.
38. From the Sketcher toolbar select **Create Centreline**.

39. Position the cursor along the vertical reference line and left-click ($X_1$) to place the first point, move the cursor down along the vertical reference line and left-click ($X_2$) to create the Centreline.

40. From the Sketcher toolbar select **Create Line** and sketch 3 lines shown in the illustration, ($X_1, X_2, X_3,$ and $X_4$). Middle-click to exit Create Line.

This is the ‘open’ profile that the revolve feature will ‘thicken’ to create a solid.

The next step is to add dimensional constraints to the sketch.

41. From the Sketch toolbar select **Create Dimension**.

The first dimension to be added will define the main diameter of the cap. The sketch only defines one side of the cap profile. In some CAD systems this would require the creation of a radius dimension, however Pro|ENGINEER allows the creation of a diameter dimension;

42. Left-click the vertical Centreline ($X_1$), then left-click the vertical sketch line ($X_2$), now left-click the Centreline again ($X_3$), then position the cursor where the dimension is to be located and middle-click ($X_4$) to create the dimensional constraint.
43. Repeat this process to add a diameter dimension to the bottle cap nozzle.

44. Add the remaining dimensional constraints, don’t worry about the sizes..

The next step is to change the dimension values.

45. From the Sketcher toolbar left-click Select Items.

46. Double left-click the dimensions and make the following changes:
   - Nozzle diameter = 10mm
   - Main cap diameter = 53mm
   - cap height = 10mm
   - cap angle = 15°
   - Nozzle height = 15mm

47. The Sketch is now complete. In the Sketcher toolbar left-click Accept Sketch to accept and exit the Sketcher.

48. The view will still be in the same orientation used for sketching. From the top toolbar select Saved View List and from the pull-down menu select either ISOMETRIC or TRIMETRIC to return to a more suitable view orientation.

Pro|ENGINEER will automatically preview the Revolve Feature. The required thickness of the cap is 2mm on the outside of the open profile.

49. In the Dashboard enter 2.00mm for the thickness and left-click Flip Direction. This is a toggle option, each left-click switches the thickness setting to one of the following:
   - Inside
   - Outside
• Symmetrical about the profile

The thickness needs to be added to the Outside of the profile.

50. To complete the Revolve feature select **Complete Feature** at the far right-hand side of the Dashboard.

51. At this point save the part. From the top toolbar select **Save File** and in the **Save** dialog select **OK**.

**Task 12: Adding detail to the cap**

The cap is a push fit and hence a ridge needs to be added to the inside of the cap to fit onto the bottle top. The ridge will be created by revolving a semi-circle around the inside of the cap to add material.

52. From the Feature toolbar (down the right-hand side of Pro|ENGINEER) select **Revolve Feature**.

Pro|ENGINEER will display the Revolve Feature tools and options in the ‘Dashboard’ along the bottom of the Pro|ENGINEER window.

A Revolve Feature is a Sketch Based Feature; the next step is to define the sketch.

53. In the Revolve Feature Dashboard left-click the word **Placement**, this will open a small Dashboard dialog, select **Define**.
54. In the Pro|ENGINEER Feature Navigator move the cursor over the FRONT datum plane and select it with a left-click. This will populate the Plane data box. Pro|ENGINEER will then automatically suggest/select the TOP datum plane as the Reference Plane to define the Sketch Orientation and Sketch view direction.

55. To accept these references and enter the Sketcher select Sketch. Once in the Sketcher Pro|ENGINEER will orient the view to look directly onto the Sketch Plane. If this doesn’t happen, in the top toolbar select to reorient the view.

Based on the selection of the FRONT datum Plane as the sketch plane and the TOP datum plane as the orientation plane, Pro|ENGINEER has automatically created two reference lines in the sketch. A further reference line is needed:

56. To make selection of the correct geometry easier; from the main toolbar select Display Hidden Lines.

57. From the Pro|ENGINEER top toolbar select Sketch and in the pull-down menu that appears select References…

58. Select the inner edge of the cap, (as shown in the illustration), and in the References menu select Close.
59. From the Sketcher toolbar select **Create Circle**, position the cursor on the newly created Reference line and left-click to place the circle centre. Drag the cursor away from the first point and left-click again to create a small circle.

60. From the Sketch toolbar select **Dynamic Trim** and select the circle at the point indicated in the illustration. Middle-click to exit Dynamic Trim. This will leave just one half of the circle.

61. From the Sketch toolbar select **Create Line** and sketch a line from one end of the semi-circle to the other (to create a closed profile).

The next step is to add the required dimensional constraints and set them to the required values.

62. From the Sketcher toolbar select **Create Dimension**. Using the techniques used to create previous dimensional constraints add a dimension between the horizontal reference line and the top end of the semi-circle. Now add the radius dimension, (there may already be a weak dimension for the radius)

63. From the Sketcher toolbar left-click **Select Items**. Double-left-click each dimension and change the values so that the circle radius is **1.5mm** and the linear dimension is **7.0mm**

64. The revolve feature needs an axis of revolution; From the Sketcher toolbar select **Create Centreline**. As done in the previous task create the centreline along the long vertical reference line.
65. The Sketch is now complete. In the Sketcher toolbar left-click **Accept Sketch ✔** to accept and exit the Sketcher.

66. To complete the Revolve feature select **Complete Feature ✔** at the far right-hand side of the Dashboard.

67. At this point save the part. From the top toolbar select **Save File** and in the **Save** dialog select **OK**.

**Homework**

Students use their understanding of Pro/ENGINEER, existing bottle designs and bottle manufacturing processes to design their own sports drink bottle that can be manufactured in school.
Lesson six – Computer model of own design

Aim:
In this lesson students build on the Pro|ENGINEER skills they have learned to model their own bottle design on computer.

Learning objectives:
By the end of this lesson students should:
° Be more aware of the possibilities and limitations of modelling 3D shapes using Pro/ENGINEER.
° Be able to combine 3D sketch based features to create a simple bottle shape of their own design.
° Be able to apply direct features to modify basic shapes.

Focus:
During this lesson students use the techniques they have learned in Pro|ENGINEER to model their own design for a sports drink bottle.
Students should have the opportunity to see the designs of other students and share the highs and lows inherent in learning new software.
Lesson seven – CNC manufacture of bottle

Aim:
Students will use the available CNC machining or rapid prototyping equipment to manufacture the shape of their bottle.

Learning objectives:
° Be aware of the running costs of operating CNC/RP equipment
° Be aware of downstream production techniques including vacuum forming, casting, moulding, etc.
° Understand the health and safety requirements of operating CNC/RP equipment
° Be able to use CNC/RP equipment under close supervision to create their own bottle design.

Focus:
The format for this lesson depends entirely on the type and quantity of manufacturing equipment the school has and the level of technical support available. An absolute minimum provision would be a small 3D engraver/mill used to produce a design in two halves that are then joined together. Schools should consider downstream manufacturing processes like vacuum forming for prototyping designs.

Homework
Create a process diagram explaining the sequence of feature creation.
Lesson eight – Complete the cap and assembly

Aim:
In this lesson students are introduced to the assembly tools in Pro|ENGINEER combining the bottle and cap.

Learning objectives:
By the end of this lesson students should:
° be aware how 3D parametric solid modelling software can create multi-part assemblies
° Understand the principles of assembly in Pro|ENGINEER and the tools and procedures to accomplish this.
° Be able to assemble components using insert, mate and align constraints in Pro|ENGINEER.

Focus:
During this lesson you will be shown the process of taking a 3D design from the screen through ‘post-processing’ software to a set of machine code which controls a computer numerically controlled machine which cuts the design out.
This will probably be a foam model which is a quick inexpensive way of proving the design and checking the machining instructions.

Task 13: Adding grip-ribs around the cap

To aid removal of the push-fit cap a series of ridges will be applied around the cap.

1. From the Feature toolbar left-click Extrude feature.

Pro|ENGINEER will display the Extrude feature tools and options in the ‘Dashboard’ along the bottom of the Pro|ENGINEER window.

An Extrude Feature is a Sketch Based Feature; the next step is to define the sketch.
2. In the Extrude Feature Dashboard left-click the word **Placement**, this will open a small Dashboard dialog, select **Define**.

Before you start sketching Pro|ENGINEER needs to know where to place the Sketch and how it is to be oriented. Pro|ENGINEER will issue a prompt along the bottom of the Pro|ENGINEER window asking you to; **Select a plane or surface to define sketch plane.**

Pro|ENGINEER will also display the Sketch dialog which captures the selection of **Sketch Plane** and **Sketch Orientation** information.

The ridge sketch needs to be placed on the bottom flat face of the cap.

3. Hold down the middle mouse button and move the cursor upwards (middle-drag), this will re-orient the view to make selection easier.

4. Select the bottom face of the cap, and in the Sketch dialog left-click **Sketch**

Once in the Sketcher Pro|ENGINEER will orient the view to look directly onto the Sketch Plane. If this doesn’t happen, in the top toolbar select ![reorient](image) to reorient the view.

Based on the selection of the cap face as the sketch plane, Pro|ENGINEER has automatically created two reference lines in the sketch. A further reference line is needed:

5. From the Pro|ENGINEER top toolbar select **Sketch** and in the pull-down menu that appears select **References...**
6. Select the outer edge of the cap and in the References menu select Close.

7. From the Sketcher toolbar select Create Circle ○.

8. Sketch the circle with its centre at the intersection of the horizontal reference line with the circle reference line (X₁), move the cursor out and create the circle with another left-click (X₂).

9. From the Sketch toolbar select Dynamic Trim and select the circle at the points indicated in the illustration. Middle-click to exit Dynamic Trim. This will leave just one half of the circle.

10. From the Sketch toolbar select Create Line and sketch a line from one end of the semi-circle to the other (to create a closed profile).

The next step is to add the required dimensional constraints and set them to the required values. Pro|ENGINEER has already created a ‘weak’ dimensional constraint for the circle’s diameter.

11. From the Sketcher toolbar left-click Select Items. Double-left-click this weak dimension and change the value to 4mm.

12. The Sketch is now complete. In the Sketcher toolbar left-click Accept Sketch to accept and exit the Sketcher.
13. The view will still be in the same orientation used for sketching. From the top toolbar select **Saved View List** and from the pull-down menu select either **ISOMETRIC** or **TRIMETRIC** to return to a more suitable view orientation.

Pro|ENGINEER will preview the Extrude feature based on default dashboard setting. For the purposes of the ridge these defaults are wrong.

14. In the dashboard select the depth option and from the slide-up menu select **Up to Selected curve, surface...**, in the graphics window select the top angled face.

15. Pro|ENGINEER now has sufficient information to create the require ridge feature. To complete the Extrude feature select **Complete Feature** at the far right-hand side of the Dashboard.

Notice how Pro|ENGINEER has created the top of the ridge so that it maintains the angle of the cap.

To create the required grip on the cap the single ridge will be copied/patterned around the cap.

16. In the Feature Tree/navigation pane right-click the Extrude feature and from the menu that appears select **Pattern**

Pro|ENGINEER will display the Pattern dashboard

```
<table>
<thead>
<tr>
<th>Dimensions</th>
<th>Table Dimensions</th>
<th>References</th>
<th>Tables</th>
<th>Options</th>
<th>Properties</th>
</tr>
</thead>
<tbody>
<tr>
<td>Dimension</td>
<td></td>
<td></td>
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</tr>
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<td></td>
<td>1</td>
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<td></td>
<td>Select items</td>
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<td></td>
<td>2</td>
<td>2</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
```

17. The default type of Pattern is Dimension, in this case the grip pattern will an Axis pattern. In the Dashboard left-click the upturned arrow next to Dimension
and, from the list of options select **Axis**. Pro|ENGINEER will prompt

> Select a Datum Axis to define the pattern center.

If there are no axes visible go to the top toolbar and toggle on the display of **Axes**.

18. Once axes are visible, left-click the axis which passes through the centre of the cap (there should only be one axis). This will change the dashboard display and also display some default pattern graphics. (The black dots indicate where each instance of the ridge will be placed)

The grip pattern requires 24 ridges to be equally spaced around the cap.

19. In the Dashboard select **Equi-Spaced**. The dashboard options will now change and the pattern preview graphics will also change.

20. In the Dashboard change the number of instances to **24** and hit the **Return** key to update the preview.

21. To accept and complete the pattern left-click **Complete Feature** from the right-hand side of the dashboard.

22. At this point save the part. From the top toolbar select **Save File** and in the **Save** dialog select **OK**.
Task 14: Assigning material properties and other parameters

While the cap ‘model’ is finished, in that it’s geometric form is complete, Pro|ENGINEER can capture more information which is an important part of the design and engineering process. This additional information includes material and project information.

23. From the Pro|ENGINEER top toolbar left-click Edit, and from the pull-down menu select Setup…

24. This will open up the Pro|ENGINEER menu-manager. Left-click Material.

25. Pro|ENGINEER will now display the Material dialog. Select LDPE (Low Density PolyEthelene), left-click the assign button followed by OK.

Pro|ENGINEER will now apply this material to the bottle solid model.

26. To close the menu-manager select Done.

The next step is to assign none geometric parameters to the cap such as Description and Project.

27. From the Pro|ENGINEER top toolbar left-click Tools and from the pull-down menu select Parameters…

28. In the Parameter dialog fill in the Values for DESCRIPTION, MODELLLED_BY and PROJECT. Click OK to accept your Parameters.
29. The cap is now finished. At this point save the part; from the top toolbar select **Save File** and in the **Save** dialog select **OK**.
Bringing both the bottle and cap together requires the creation of an Assembly.

**Task 15: Creating the bottle assembly in Pro/ENGINEER**

30. From the Pro/ENGINEER top toolbar left-click **Create New File**. In the dialog box that appears enter “bottle_assembly”.

31. Notice the default **Type** is currently set to Part, this needs to be set to Assembly, simply left-click the word **Assembly**.

32. Left-click **OK** to accept the settings and create the new Pro/ENGINEER assembly file.

When the part opens you should see the default Datum Planes, **ASM_FRONT**, **ASM_TOP**, & **ASM_RIGHT**, and the default coordinate system **ASM_DEFAULT_CSYS** displayed in the graphics window and Feature Navigation pane.

For the purposes of this activity the **ASM_DEFAULT_CSYS** is not required;

33. From the Pro/ENGINEER top toolbar left-click **Coordinate System on/off** to turn off the display.

**Task 16: Adding the bottle part to the assembly**

34. From the **Assembly** toolbar select **Add Component**.

35. In the **Open** component dialog select **bottle_prt** then left-click **Open**.

Pro/ENGINEER will preview the bottle in the assembly and prompt you to **Select any reference for auto type constraining**.
The Assembly Dashboard will also be displayed along the bottom of the Pro\ENGINEER window.

As this is the first component to be added to the assembly its placement will use the Default option.

36. In the Dashboard left-click the word Automatic and from the slide-up menu that appears select Default.

37. The bottle is now fully constrained within the assembly. From the right-hand side of the dashboard select Accept ✓ to finish placement of this component.

Before adding the cap the display of Datum Planes can be turned off.

38. From the top toolbar select Display Datum Planes ✗ to toggle off their display

**Task 17: Adding cap to the bottle**

39. From the Assembly toolbar select Add Component

40. In the Open component dialog select cap_prt then left-click Open.

Pro\ENGINEER will display the Assembly dashboard and preview the cap in the assembly and prompt you to Select any reference for auto type constraining.
Pro/ENGINEER is asking for matching pairs of geometry to define the required assembly constraints.

41. To aid assembly the cap can be positioned closer to its point of assembly. Hold down the Ctrl and Alt keys and then right-drag the mouse to move the cap. Place the cap near the top of the bottle and release all the keys. When the cap is close to the bottle top release the keys and mouse button.

42. Move the cursor over the cylindrical face of the bottle neck, Pro/ENGINEER will pre-highlight the ‘face’, left-click to select it, then move the cursor over the cylindrical face of the cap nozzle and when the face is pre-highlighted left-click to select it.

Pro/ENGINEER will now automatically create an Insert assembly constraint and move the cap accordingly.

43. Now move the cursor over the top flat face of the bottle until it pre-highlights then left-click to select it. Then using middle-drag re-orient the view so you can see underneath the cap, now move the cursor over the small bottom face of the cap, when it pre-highlights left-click to select it.

Pro/ENGINEER will most likely create a coincident Mate assembly constraint. While the Mate constraint is correct the cap should be offset rather than coincident.
44. In the Assembly dashboard click the small up-turned arrow next to the **Coincident** option to open the slide-out menu, from this menu select the **Offset** option.

45. In the dashboard enter an offset value of **5.5mm**.

![Offset Value](image)

46. Pro|ENGINEER now has sufficient information to fully constrain the cap onto the bottle. From the left-hand side of the dashboard select **Accept** to finish placement of this component.

47. At this point save the part. From the top toolbar select **Save File** and in the **Save** dialog select **OK**.

You have now successfully finished the modelling and assembly of a Sports Drink Bottle.
Lesson nine – Engineering drawing

Aim:
In this lesson students are taught how to use Pro|ENGINEER to create an engineering drawing for their bottle design.

Learning objectives:
By the end of this lesson students should:
- Be aware of the international standards for engineering and technical drawings.
- Understand how technical drawings are used for quality control, assembly and operation of products.
- Be able to use Pro|ENGINEER to create an orthographic drawing of their bottle including a pictorial view.

An effective method of communicating designs to other people is via the use of drawings. Pro|ENGINEER allows Designers and Engineers to quickly produce engineering production drawings directly from the solid model.

Paper drawings have been the traditional method of communicating product design information to manufacturing but the use of solid modelling has allowed a more direct and automated link.

Using drawings requires the manufacturing engineers to interpret 2D orthogonal views where as the 3D solid model contains more information and is easier to visualise. The use of Computer Numerically Controlled (CNC) machines now allows engineers to produce components directly from the solid model.

Using of this level of automation means that drawings are being used to provide overall dimensions and inspection information.

In this section you will learn how to produce a detail drawing of the water bottle.

Homework
Students begin collating all their information into an e-folio using PowerPoint.
Task 18: Creating the drawing

1. From the Pro|ENGINEER top toolbar left-click Create New File. In the dialog box that appears the default Type is Part, left-click Drawing. Enter “bottle” for the Name.

2. Notice the “Use default template” option is checked. To learn how drawing views are created in Pro|ENGINEER drawings we will NOT accept this default option. Left-click the small tick to deselect this option.

3. Left-click OK.

Note: Using the default template will automatically create views which have been pre-defined within the default template file.

4. In the New Drawing dialog that appears Pro|ENGINEER is giving you the option of selecting which model is to be used within the drawing. Make sure the Default Model is the BOTTLE_ASSEMBLY.ASM.

5. In the Specify template section select Empty with format, this will change the options in the Format section of the dialog.

6. For the Format make sure A3_FORMAT is selected. If not, use the Browse option to browse to the drawing template directory, select A3_FORMAT and click Open.

7. Left-click OK to accept these settings and create the drawing.
8. Pro|ENGINEER will create an A3 drawing with drawing border/format. 

Notice that the User Interface has changed to display the commands and options relevant to Drawing creation.

The first step is to create the initial / general view.

9. From the Drawing toolbar select **Create a general view**.

10. Pro|ENGINEER may display a small dialog/menu. Select the **No Combined State** option and left-click **OK**. If this dialog does not appear go straight to the next step.

11. Position the cursor in the top left quadrant of the drawing and left-click to position the centre of the first view.
Pro|ENGINEER will also open up the Drawing View dialog.

12. In the Model view names section select FRONT and then left click **Apply**.

This will orient the newly created view to match the FRONT view as defined in the Bottle Assembly.

The next step is to change the view display.

13. In the Categories section of the Drawing View dialog select View Display.

14. Change the Display Style from Follow Environment to No Hidden and then left-click **Apply** followed by **Close**.

The view may also display the Datum Planes and Axes; left-click Datum Plane Display on/off and Coordinate systems on/off.

15. To adjust the position of this initial view, make sure the view is selected, place the cursor over the view and hold down the right mouse button. In the floating menu left-click Lock View Movement.

This will toggle the option which either locks the position of the view or allows the view to be moved.
16. Now that view movement is unlocked, re-select the view and using left-click-drag you can move the view to the required location.

The next step is to create what are known as Projected Views.

17. Left-click the initial view (a red box will appear around the view to indicate it is selected), then right-click and hold to open a floating menu.

18. Select **Insert Projection View**…

19. Now move the cursor to the right of the initial view. Pro|ENGINEER will display a green rectangle previewing where the view extents lie.

   Notice that the movement of the projected view is locked to the horizontal. If you move the cursor below the initial view the preview is locked to the vertical.

20. Position the preview to the right of the initial view and Left-click to place the new view.
21. Now create another projected view, this time place it below the initial view.

22. Change the Display Style of these two new views to No Hidden (see step 13, double-left-click each view in turn to open up the Drawing View dialog)

23. At this point save the drawing. From the top toolbar select Save File and in the Save dialog select OK.

Now that the views have been created we can add dimensions and annotations to the drawing.

24. From the Drawing toolbar select Show/Erase. This will open the Show/Erase dialog box.

The first step is to show the centre lines for the bottle.

25. In the Type section of the Show/Erase dialog left-click Axis. Note: Selecting the options in the Type section toggles on/off their selection.

26. In the Show By section make sure Feature is selected.
27. Move the cursor over the initial view until the main body of the bottle pre-highlights, left-click to select the bottle feature \((X_1)\) and Pro|ENGINEER will create two centre-lines indicating the axes of the bottle extrude feature.

28. In the smaller Select dialog box select \(\text{OK}\). This informs Pro|ENGINEER you have finished your selections and will change the options in the Show/Erase dialog.

29. In the Show/Erase dialog select \(\text{Accept All}\), followed by \(\text{Close}\). Pro|ENGINEER will have created centre-lines in all 3 views.

To improve the aesthetics of the newly created centre-lines the length of the centre-line can be manually adjusted.
30. In the lower view select one of the newly created centre-lines.
The centre-lines will now have drag handles at each end.

31. Using these drag handles drag the centre-lines to the required lengths, and repeat this process for the other centre-lines in this and the other views.

The next step is to create dimensions.

32. From the Drawing toolbar select Show/Erase. This will open the Show/Erase dialog box.

33. In the Type section select Axis (to toggle off this option) and then select Dimension.
34. Move the cursor over the initial view until the main body of the bottle pre-highlights, left-click to select the bottle feature \(X_1\) and Pro|ENGINEER will preview all the dimensions used to create the main bottle extrude feature in two views.

35. In the smaller Select dialog box select \(\text{OK}\). This informs Pro|ENGINEER you have finished your selections and will change the options in the Show/Erase dialog.

36. In the Show/Erase dialog select \(\text{Accept All}\), followed by \(\text{Close}\).

Pro|ENGINEER will now show all the dimensions.

37. All the newly created dimensions will still be shown in red indicating they are still selected. Left-click anywhere in the graphics window to de-select these dimensions.

38. If the position of the dimensions is not quite as you require select each dimension in turn and drag the dimension to the required location and use the drag handles to adjust the dimension leaders.

39. You can remove any dimension by selecting it and then right-click to bring up the pull down menu and select \(\text{Erase}\).

All the dimensions created up to now have been created by 'showing' the dimensions used in the creation of the feature within the model. The next step is to create a new dimension.
40. From the Drawing toolbar select Create **Standard Dimension**.

41. In the initial view left-click to select the top of the Cap and the bottom of the bottle ($X_1, X_2$) and then middle-click to position the dimension ($X_3$).

42. This introduction to drawing is now complete. At this point save the drawing. From the top toolbar select **Save File** and in the Save dialog select **OK**.
Lesson ten/eleven – Finishing bottle prototype

Aim:
Students use these two lessons in the workshop to finely finish the surface of their prototype bottle and apply information/surface designs.

Learning objectives:
By the end of this lesson students should:
° Be aware of a range of prototype finishing techniques.
° Understand the role of handling prototypes in product development.
° Be able to produce a well finished bottle prototype as a proof of concept design.

This lesson will involve hand finishing the foam CNC model in a workshop ready for your final presentation. You must make sure you are very clear about the health & safety implications for working with foam and take all necessary precautions to keep yourself and others safe and healthy.

Homework
Students complete their e-folio for presentation in final lesson.

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