Ball Valve Assembly

Supplied are the main components of a ball valve. In this exercise you will assemble the valve as shown below Left. (N.B. Socket head cap screws are not supplied…these will be created later using the Solid-Works toolbox).

On completion of the assembly, we will create the exploded view as shown on the right.

Downloading Parts
Copy the folder from ‘Valve parts’ from S:\classes\sean-dalton\ and paste it into CAD folder on your H: drive.
Rename this folder from ‘Valve parts’ to ‘Valve-assembly’.

Assembly
Start first of all by opening the main body component.
Next, transfer this component into an assembly by selecting ‘Make Assembly from Part’.
Choose the assembly template when prompted.

Assemble the remaining parts as shown.

To insert components into the assembly resize the SolidWorks window to occupy half the screen and windows explorer, the other half.

Drag and drop the appropriate number of each component into the assembly and apply assembly relationships as required.

Note that the socket head cap screws are not supplied. These will be created in the next section using the SolidWorks Toolbox. The SolidWorks toolbox is an add-in utility which is used to create standard engineering components.
Clipping planes
The model may be examined at any time using the clipping plane tool.
To activate select a plane followed by the clipping plane tool.

Once activated plane may be moved or rotated as required.
The clipping plane is a visualisation tool rather than a geometric feature.

Toolbox
The Solid Works toolbox will now be used to create the cap screw.
First of all ensure that the SolidWorks toolbox is activate by selecting the (+) adjacent to the word Toolbox.
If the (+) is not present it is not active.
To activate the toolbox choose Tools Add-ins and select toolbox browser.

For the Socket Head cap screws in this exercise, use ISO 4762 M10 x 25.
You only need to create one. To do this:
• Select the library symbol
• Expand toolbox
• Expand ISO
• Expand bolts and screws
• Select Hex socket head cap screws

Finally right click on ‘Hex Socket Head ISO 4762’ and select Create Part…

In the dialog which appears specify a size of M10 and a length of 25 followed by accept.

Finally save this component in the folder with all the other valve components using the name M10-25-SHCS
Finally insert this component into the assembly as many times as is required.
Assembly cut-out
When working with assemblies it can sometimes be difficult to appreciate the internal detail fully. It can be useful to create an assembly cutout. This is similar to creating a cut out in part mode but has additional benefit of being able cut several components simultaneously.

Creating an assembly cutout
To create an assembly cut first of select the front plane and draw a rectangle from the pipe centre the upwards and to the right.

Once complete, select the extruded cut, cutting through all in both directions.

Once satisfied that the assembly cut has worked correctly, suppress this feature to return to the original 3d model.

Generally assembly cut-outs should only be used to help visualisation etc and should be suppressed afterwards.

Another way for doing this is to use a clipping plane as illustrated in the previous exercise.
Exploded view

To create an exploded view select insert Exploded view. The top section is where the explode steps are listed. At first this will be empty but, as the explode sequence is developed each new step will be added to this section.

**Explode Step 1**

For the first step you will move the left hand flange and its associated screws to the left by **50mm**.

To do this, select the Left flange and the 4 screws (all selected items appear in section A below).

Next select the blue arrow shown and manually drag to the left. Finally overwrite the value of distance in the and specify a value of **50mm**.

*(If it is required to adjust the value enter a new value in the distance box and select [Apply] to preview the effect).*

When complete select [Done] to complete the step.

**Explode Step 2**

Next create another explode step by move the screw by an additional **40mm**.
**Explode Steps 3 / 4**

Again repeat this process on the left hand side moving the right hand flange and corresponding set screw by **100mm** and **40mm** respectively.

(The larger distance to make room for ball).

**Explode Steps 5 / 6 / 7 / 8 and 9**

In the next steps we will explode the upper components in a natural interactive.

Drag all of the upper components in a logical sequence to an appropriate distance above the main body.

If the distances do not work out correctly they can be manually adjusted later.

The finish exploded view should look as shown below.

Finally collapse the assembly before proceeding.
**Accessing and editing the exploded view**

To access the explode view details select the configuration tab at the top of the feature manager. This gives the user access to each individual step of the explode view.

**Expanding and collapsing an Exploded view**

On completion of the exploded view the user can expand explode the assembly instantaneously or the user can animate the explode operation. Likewise if the assembly is already exploded the user may collapse the assembly instantaneously or may choose to animate it.

On animating the explode operation the following Animation controller toolbar will appear. When finished select the [X] button to remove the.

**Editing an Exploded view**

To edit an exploded view after it has been created right click on the exploded view name and select Edit Feature.

To edit an individual step, right click on the step in question and select Edit Feature.
**Ball Valve assembly draft**

Create a new drawing using the standard drawing template.

**Creating additional sheets**
Create a second sheet for use later.
To create a second sheet,
- select the Sheet1 tab,
- press the CTRL key
- then drag mouse and release

This will create another sheet with the same border as in sheet one. This will be used later for creating isometric and exploded views.

**Drafting the Valve assembly**
On sheet1 create the following views:
- Elevation
- Plan
- End view
- Isometric view.
Create these views in the orientations shown.
Next you will create a half section of the end view.

**Half Section (broken out section)**
To create a half section of the end view. To do this:
- Use the sketching tools to draw the rectangle (A) shown.
- Exit sketch mode.
- Next select the rectangle followed by the broken out section icon. (below).
- When prompted select ‘Auto-hatching’ to ensure that the hatching lines alternate.

For the depth specify a distance of **65mm**.

65mm represents the distance from the end as illustrated in view B.
Section only
Occasionally it can be desirable to show only the cut surface detail.

This eliminates the background detail by not show any geometry behind the cutting plane.

Local sections
In a similar manner the broken out section tool can be used to create a local section of a specific detail. Take for example the bolts holding the flanges to the main body.

This time issue the local section (broken out section) command first. This will result in the spline tool being active by default. Sketch a closed spline around the detail in question and specify the appropriate depth. Finally turn off sectioning in the screw.

Isometric and exploded views
On a separate sheet but within the same draft document insert two isometric views of the assembly. Copy and paste from Sheet1 if required.

Next right mouse click on the left most view and select properties. In the dialog box which appears select ‘Show in exploded state’.

View settings
Generally speaking, the following setting should be used in draft views. Regular views Turn off tangent edges, Turn on Hidden detail. Isometric views Turn off hidden detail, Turn on tangent edges with font.